Wound Composite Modeler For Abaqus

Abaqus Version 6.7-3 © 2007 SIMULIA, Inc.

User's Manual

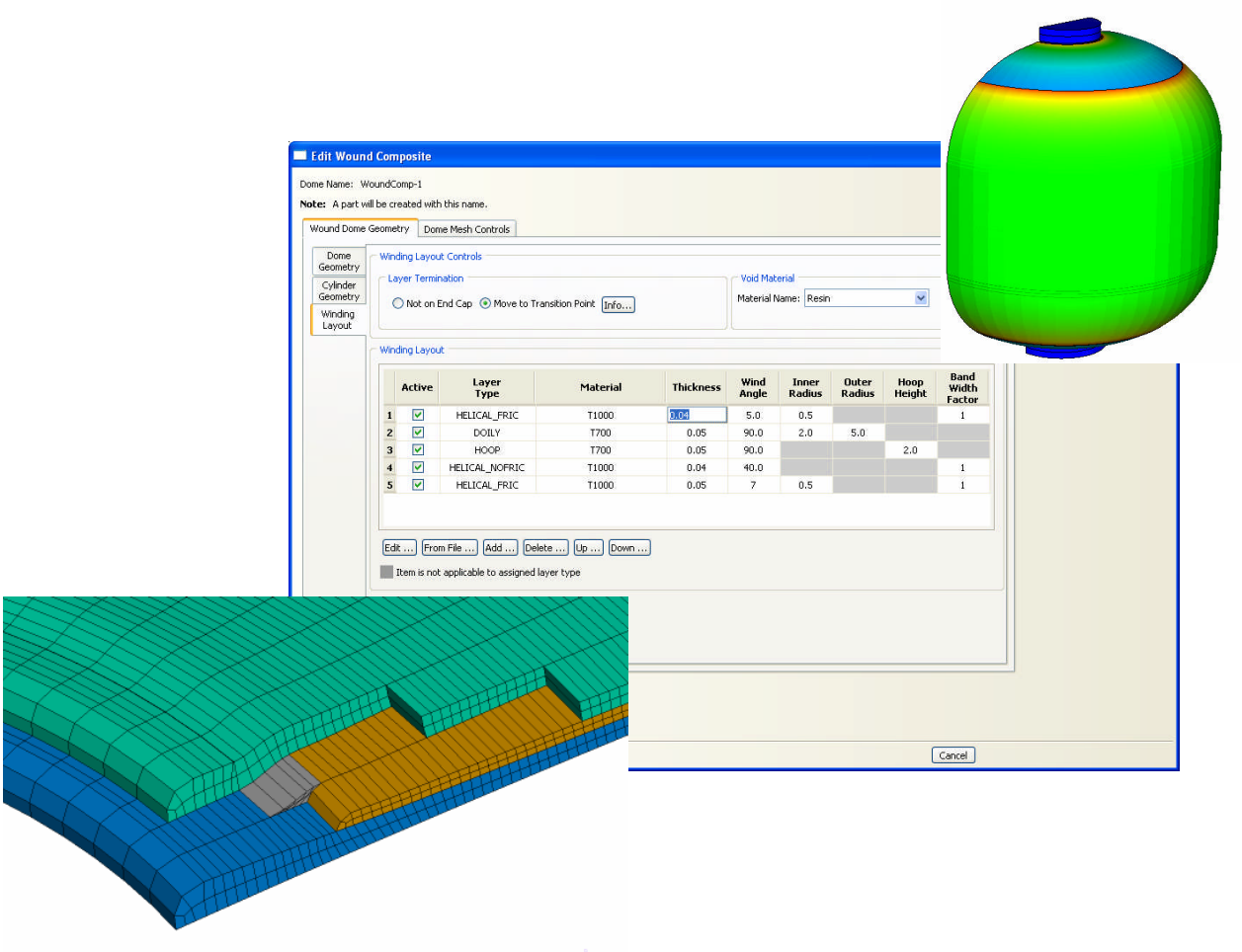


Table of Contents

1. Background 3
   1.1 Plug-in Overview 3
2. Underlying Dome Geometry 5
   2.1 Geodesic Dome 5
   2.2 Elliptical, Spherical Domes 7
   2.3 User-Defined Dome 7
   2.4 From-Part Instance Dome 8
   2.5 Cylinder Geometry 9
3. Winding Layout 9
   3.1 Helical Layers 9
      3.1.1 Helical Layer Thickness Buildup 10
      3.1.2 Helical Shear Ply 11
      3.1.3 Helical Layer with Friction 11
   3.2 Doily Layer 12
   3.3 Hoop Layer 12
   3.4 Void Creation 13
   3.5 Layer Termination Options 14
   3.6 Layer-Level Controls 15
      3.6.1 End Controls 16
      3.6.2 Layer-Level Mesh Controls 18
      3.6.3 CAD Data Fitting 19
   3.7 Dome-Level Mesh Controls 21
4. Wound Composite Tank Creation 23
   4.1 Overview 23
      4.1.1 Axisymmetric Continuum Geometry 24
      4.1.2 3D Continuum Geometry 24
      4.1.3 3D Shell Geometry 26
      4.1.4 Axisymmetric Shell Geometry 28
      4.1.5 Continuum Shell Geometry 28
   4.2 Dome Assignments 31
   4.3 Section Assignments 31
      4.3.1 Material Properties 32
   4.4 Load and Boundary Assignments 33
   4.5 Mesh Creation 34
   4.6 Job Creation 34
      4.6.1 Include Files 34
      4.6.2 Uvarm Subroutine 35
      4.7 Externally-Made Changes to Model 36
5. Model Creation from CAD Import 37
   5.1 Sketch of CAD Data 37
   5.2 Import of CAD Data 37
      5.2.1 Band Width Settings 38
      5.2.2 Ends Abutting Polar Boss 39
      5.2.3 Tank Creation 39
      5.2.4 Tank Geometry Validation 39
      5.2.5 Diagnosing Problems with CAD Import 40
6. Output Processing 40
7. Sample Test Cases 43
8. Installation of Plug-in 44
9. References 45
   Papers 45
   Abaqus References 45

1. Background

The process of filament winding has become a popular technique in a wide variety of industries for creating extremely high stiffness-to-weight structures. Aerospace industry applications include rocket propellant tanks and solid rocket motors casings. Automotive industry applications include high pressure fuel storage tanks for hydrogen powered automobiles.

The difficulty in accurately analyzing the behavior of filament wound structures derives from the varying orientation of the wound filaments throughout the structure. The standard capabilities of commercial finite element codes are inadequate to model the variation of fiber orientation in a practical way. Thus, the Wound Composite plug-in for Abaqus was warranted. This Wound Composite Modeler for Abaqus plug-in was developed to analyze a wide variety of axisymmetric or three-dimensional, wound composite pressure vessels. This documentation describes the capabilities and usage of the plug-in.

Figure 1: Composite Overwrapped Pressure Vessel

1.1 Plug-in Overview

The Wound Composite Modeler for Abaqus plug-in is a vertical application which is designed to facilitate the creation of an entire axisymmetric or three-dimensional finite element model of a composite overwrapped pressure vessel (COPV). The model created may consist of a single dome and thus a symmetry plane, or consist of a top and bottom dome which provides for a full representation of the tank. The plug-in automates the creation of the tank geometry and its corresponding mesh, element-by-element material property assignments, load and boundary condition assignments. The resulting model will be run-ready – all from a single set of dialog boxes.

The process of creating a COPV model consists primarily of two steps. The first step involves creating one or more wound composite domes. The wound composite domes are essentially axisymmetric templates of the top or bottom of the tank and are used to create the full tank. The second step involves selecting the domes to combine into a single axisymmetric or three-dimensional tank model, then selecting the loads and boundary conditions to be applied to the tank.

Creating a wound composite dome begins by selecting the **Plug-Ins->WoundComposites->Manager** menu item which then brings up the one of the two major dialogs of the Wound Composite Modeler: the *Wound Composite Manager* dialog. From this first dialog selecting the **CREATE** button from the **TANK GEOMETRY** tab brings up the second major dialog: the *Wound Composite Dome* dialog. From this dialog, any axisymmetrically-shaped dome with a cylinder attached can be generated. A toggle is supplied to skip the time consuming step of generating a mesh, thus reducing the time required for each iteration on the

geometry. Once satisfactory dome geometry has been created, then the toggle for mesh creation can be turned back on and iterations on the mesh can then be performed.

Creation of a dome results in the creation of a part within Abaqus/CAE. However, this part is not instanced into the model. This part serves as a guideline for the creation of a portion of the tank and may be one of a library of domes.

The final tank model is created by choosing a top dome, and possibly a bottom dome, from the library of domes, then combining them to make the final tank model. The domes to be used are selected from the *Wound Composite Manager* dialog. Creation of the model is performed upon submission of the dialog. The tank loads and boundary conditions are also specified from the *Wound Composite Manager* dialog. Just as is the case with the creation of the domes, the creation of the entire tank may require more than a single iteration. For this reason, toggles are added to turn off the creation of the mesh and the element-by-element material property assignments since these two operations take the bulk of the model creation time. Once satisfactory tank model geometry is generated, then the mesh toggle may be turned on and iterations of the mesh can be performed. After obtaining a satisfactory mesh, then the **Regenerate Properties** button is pressed the a full, run-ready model is created.

Alterations to the model can be made within Abaqus/CAE (outside of the plug-in). Some guidelines are provided in *Section 4.6.3* on making changes to the model created by the plug-in without invalidating it.

2. Underlying Dome Geometry

A number of different underlying dome geometries are available from the **Dome Geometry** tab of the plug-in. While the shapes may be different, all may have a cylinder attached and all have a helical winding pattern described by the following equation:

$$\theta(R) = \sin^{-1}\left(\frac{R_0}{R}\right) \pm \delta\left(\frac{R - R_0}{R_{tl} - R_0}\right)^n$$

Equation 2.1: Wing Angle Definition

Here, R is the radial distance from the center line to a point in the layer, R_o is the radial distance from the centerline to the turnaround point, and R_{tl} is the radius at the dome-cylinder tangent line. A frictionless winding pattern is obtained by choosing $\delta = 0$. If friction is accounted for, $\delta \neq 0$, then the wind angle distribution will vary as described in later sections.

2.1 Geodesic Dome

The geodesic-shaped dome is a specially shaped dome which results in a constant strain state along much of the fiber as it wraps around the dome. The shape of the dome is directly related to the vessel radius and the wind angle of the layer at the tangent line.

The wind angle defining the geodesic dome shape is specified in one of two ways. Firstly, a base wind angle may be specified directly from the **Dome Geometry** tab as shown in the figure below. Secondly, if the base wind angle has not been specified, then the wind angle of the first helical layer defined under the **Winding Layout** tab is used to determine the geodesic shape.

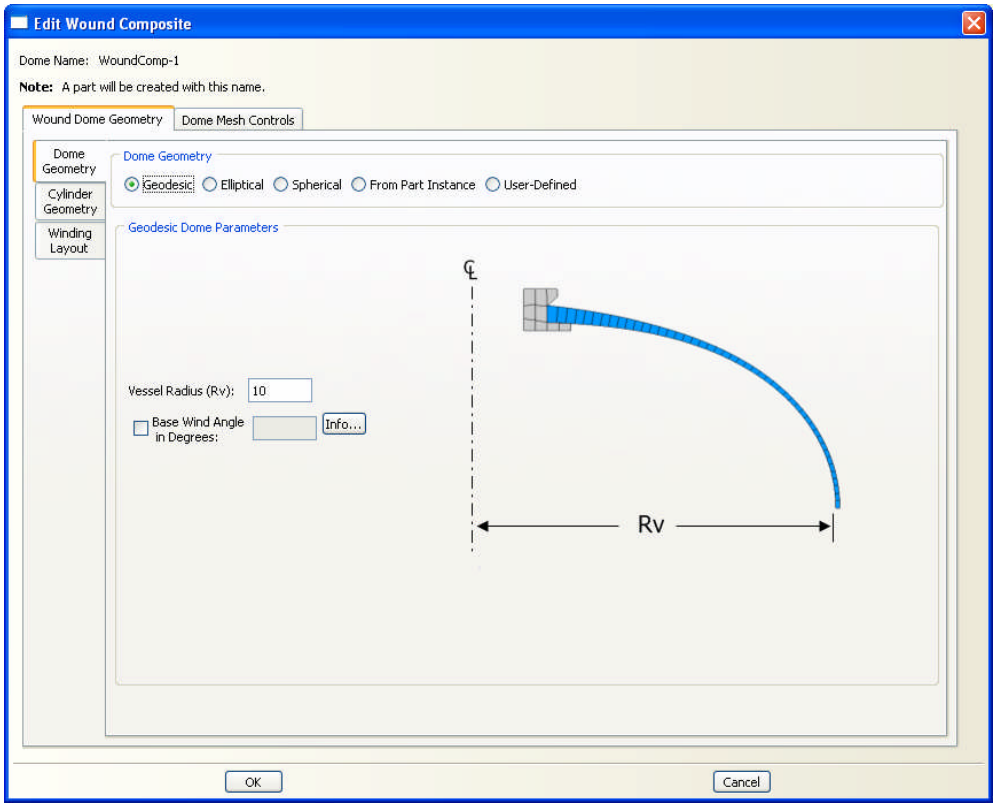


Figure 2.1-1: Geodesic Dome Input

For example, *Figure 2.1-2* shows a dome in blue which is constructed from a base wind angle of five degrees even though the first layer has a wind angle of ten degrees. Had the base wind angle not been specified, the ten degree wind angle would have resulted in the shape of the gray dome.

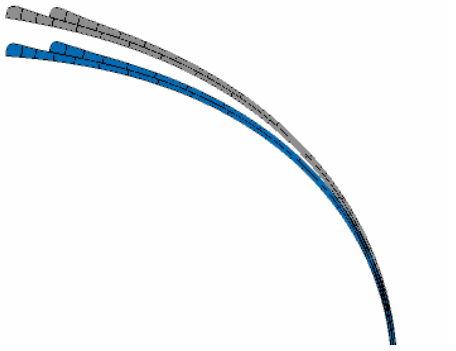


Figure 2.1-2: Geodesic Dome with/without Base Wind Angle

For a given wind angle, the helical turns around at a radius equal to the vessel radius at the tangent line times the sine of the wind angle. For a radius less than this so called turnaround radius, the shape of the dome is extrapolated out linearly from the slope at the turnaround radius. The figure below shows what this might look like.

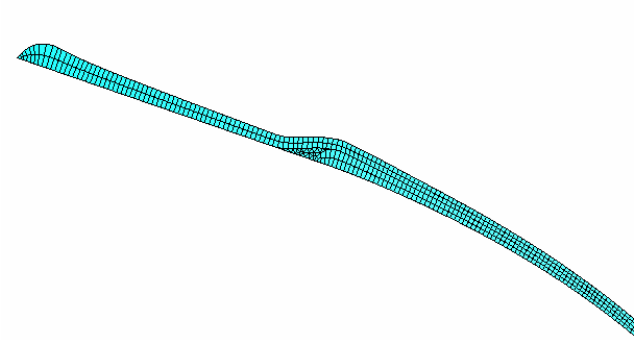


Figure 2.1-3: Layer Extending Beyond Geodesic Turnaround Radius

2.2 Elliptical, Spherical Domes

Creation of elliptically and spherically-shaped domes is available by specifying the vessel radius at the tangent line, and in the case of the elliptical dome, the radius of the ellipse in the y-direction (minor axis).

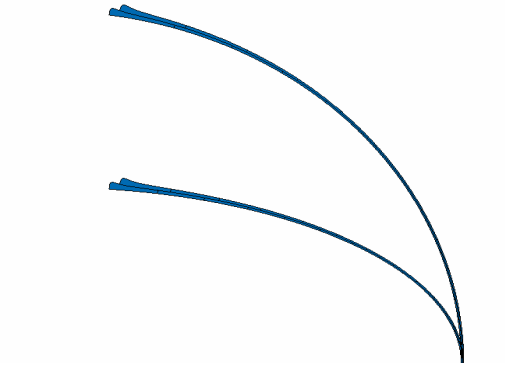


Figure 2.2: Elliptical and Spherical Domes

2.3 User-Defined Dome

Generally-shaped domes may be defined by providing a list of x-y coordinate pairs outlining the shape of the dome. A portion of the dialog is shown in *Figure 2.3*. Right clicking on the mouse while the cursor is positioned in the table will bring up a list of options which includes the filling of the table by reading from a file. The text file must contain two columns of numbers separated by either a comma or a blank space. The coordinate pairs may be listed in either ascending or descending order based on radius or position along the tank axis. The plug-in will automatically convert the order of the coordinate pairs in order to create a top (or concave) dome. When the domes are chosen as the top or bottom of the tank, then the plug-in will flip the dome geometry if necessary.

The option of connecting the discrete points by straight lines or by cubic splines is provided. Generally using splines will result in smoother shaped geometries.

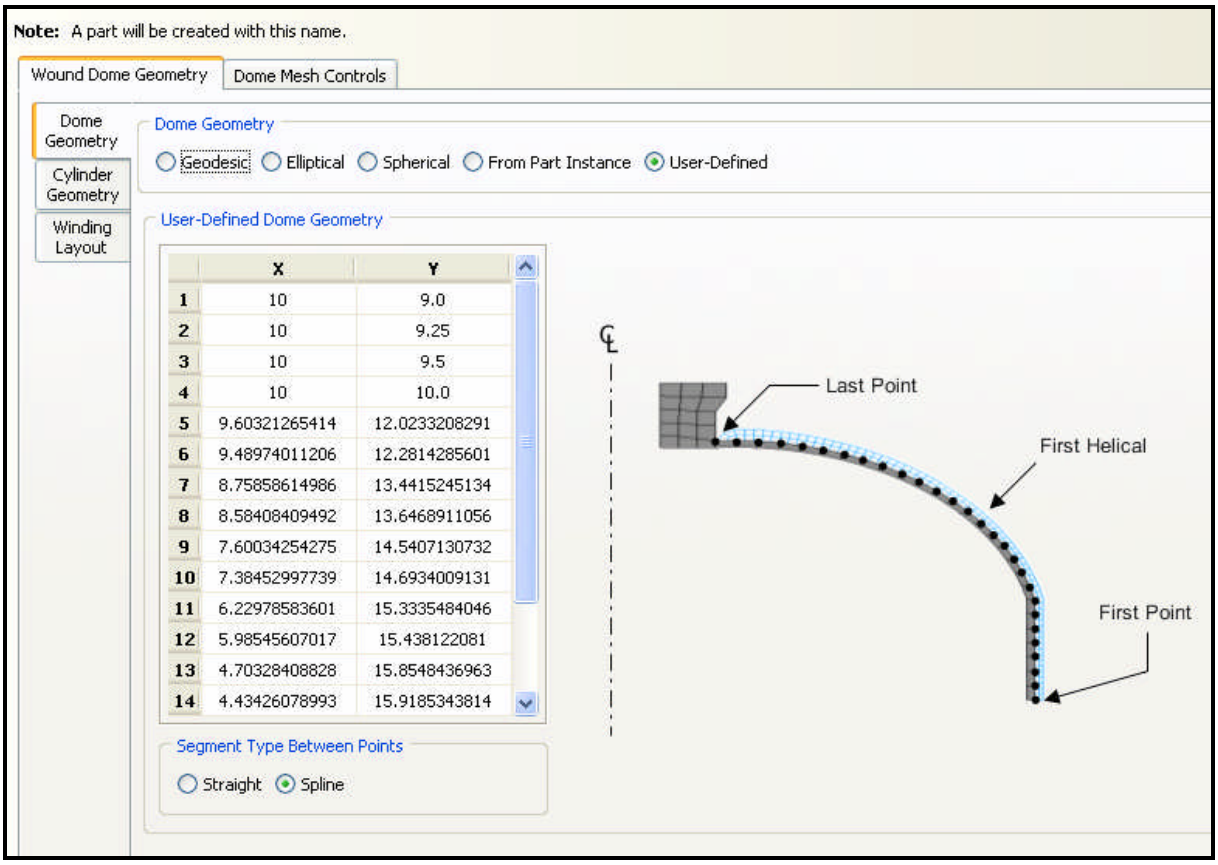


Figure 2.3: User-Defined Dome Input

2.4 From-Part Instance Dome

Wound composite layers can be made to wrap over an existing part instance that has been meshed within Abaqus/CAE. The part instance must be meshed, but may have been created from an imported orphan mesh, native geometry, or imported geometry. The points of the underlying wound composite dome geometry are extracted from the nodal coordinates along the outer surface of the part instance.

In order to define the outer surface which may be overwrapped, the user is required to generate a node set. In the case of a three dimensional model, this node set should be in the x-y plane with the axis of the tank in the global y-direction. The plug-in then generates the dome geometry from all of the nodes between the first and last nodes. The option of connecting the points using straight lines segments or splines is available.

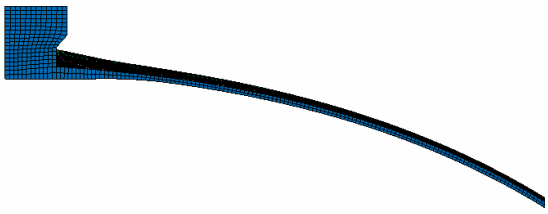


Figure 2.4: Helical Layers on Part Instance

2.5 Cylinder Geometry

A cylinder may be added to the top and/or bottom domes on any of the dome geometries. The wind angle of helical layers remains constant over the length of the cylinder.

If there are no hoops that terminate along the length of the cylinder, then all of the elements in a layer along the cylinder will be assigned a single material property and orientation. If a hoop terminates on the cylinder, then separate orientations are assigned to each element of each layer on the cylinder.

If the liner geometry is defined from an existing part which contains a cylinder length, then cylinder of the wound composite layers can be defined in one of two ways. First, the cylinder geometry may be included in the dome by selecting the start point to include the cylinder and the end point at the polar boss or turnaround radius. Secondly, the node set can be defined on only the dome with the cylinder being generated by the plug-in by manually specifying a cylinder length.

3. Winding Layout

The dialog box of the **WINDING LAYOUT** tab is shown in *Figure 3.1*. Each layer is assigned a layer type, material, thickness, and wind angle. The inner and outer radius, hoop height and band width factor are applicable to specific layer types. A row of buttons is listed below to assist in table manipulation. The **Edit** button brings up another dialog which allows detailed controls to be applied to each layer as will be discussed later. The ***Add*** button creates a new layer which is a copy the highlighted layer, including all of its layer controls information. The ***Move Up*** and ***Move Down*** buttons simply move the entire layer, along with its control information, up or down. Multiple layers can be moved up and down simultaneously.

The ***From-File*** button allows the layout to be read from a file. The text file to be read may have any number of rows and columns with the columns separated by spaces or commas. The first column of the text file, corresponding to the Active column of the table, must use either True or False. A warning will be printed if the material specified in the third column is not active in the model. Columns 10 and beyond of the text file can be used to specify the controls of each layer. To determine the type of format for these columns, select a dome and click on the ***Query*** button from the Tank Manager dialog. This not only prints a summary of the wound composite dome attributes to the command line interface, but also writes a text file named *wound_composite_dome_name.txt* which contains the winding layout of the dome in the format required for reading back into the winding layout table.

3.1 Helical Layers

The helical layer types (**HELICAL_FRIC, HELICAL_NOFRIC**) of a wound composite are typically wound in such a manner as to produce an axisymmetric layup. In other words, for every helical band at + theta, there is a corresponding band at – theta to cause the overall laminate to be a balanced angle-ply (+/- theta) laminate. This assumption is implicit in an axisymmetric model. Therefore, the wound composite plug-in requires only a single orientation angle to be given for each layer at the tangent line and the plug-in will calculate the angle-ply laminate material properties for each element within each helical layer.

Two types of helicals are available: with and without friction. The wind angle distribution of both types of helicals is described by *Equation 2.1*.

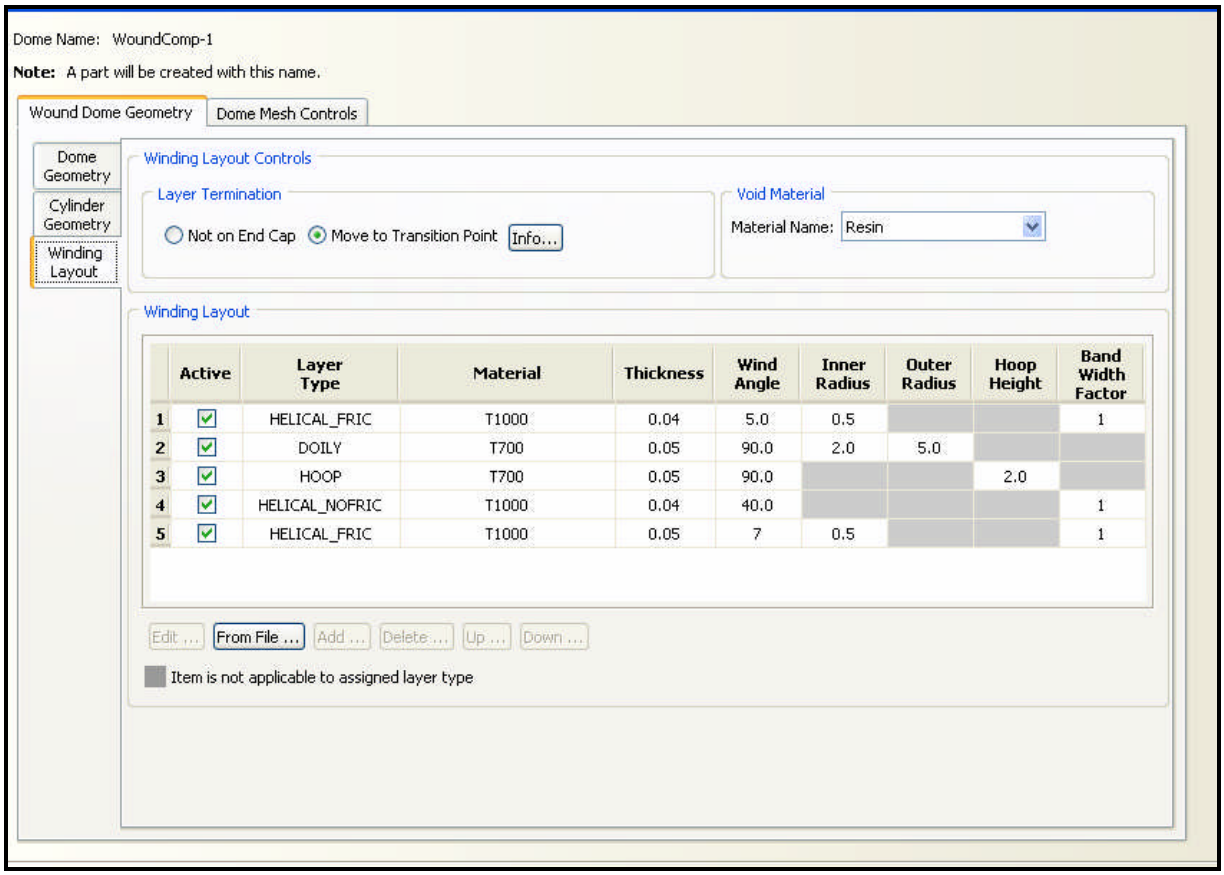


Figure 3.1: Winding Layout Table

3.1.1 Helical Layer Thickness Buildup

The thickness specified in the winding layout table is the thickness at the cylinder tangent line. As the layer traverses the dome to the polar boss, the thickness of the helical layer gradually builds up as described by *Equation 3.1.1*.

$$t(r) = \frac{t_{tl} \cdot \cos(\theta_{tl})}{\cos(\theta_r)} \cdot \frac{r_{tl}}{r + 2 \cdot BW \cdot \left(\frac{r_{tl} - r}{r_{tl} - r_0} \right)^4}$$

$t_{tl} =$ thickness at the tangent line

$\theta_{tl} =$ wind angle at the tangent line

$\theta_r =$ wind angle at radius

$r_{tl} =$ radius at the dome - cylinder tangent line

$r_0 =$ radius at the helical turnaround point

$BW =$ helical band width

$$\theta_r = \theta(r) = \sin^{-1}\left(\frac{r_0}{r}\right) \pm \delta\left(\frac{r - r_0}{r_{tl} - r_0}\right)^n$$

Equations 3.1.1

3.1.2 Helical Shear Ply

Often, a rubber shear ply is applied over the liner of the pressure vessel to accommodate the shear strain between the vessel and first filament wound layer. The shear ply can be modeled by defining a helical layer with friction specifying its terminating inner radius, and setting the band width to zero. If the layer is assigned an isotropic material, then the wind angle calculations are skipped and the layer is simply assigned the isotropic material properties.

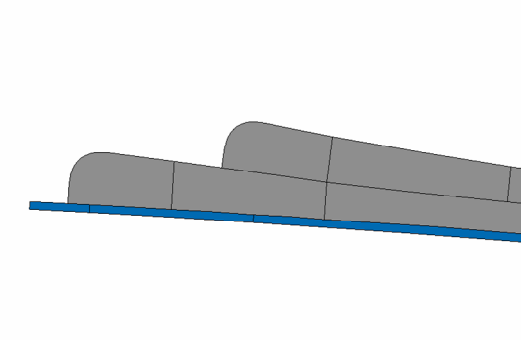


Figure 3.1.2: Zero-Bandwidth Shear-Ply

3.1.3 Helical Layer with Friction

A helical layer with-friction (**HELICAL_FRIC**) requires the wind angle and turnaround radius be specified. The wind angle distribution is determined from of *Equation 2.1* with the value of *n* set to 1.0. This equation essentially bounds the with-friction curve between two frictionless curves. The first curve is frictionless curve based on the wind angle given. The second curve is the frictionless curve based on a wind angle that would reproduce the turnaround given. The difference in wind angles (δ) between the two curves is simply ramped down linearly (for n=1.0) over the length of the layer with the turnaround specified by the user. The resulting with-friction wind angle curve is shown in red in *Figure 3.1.3*.

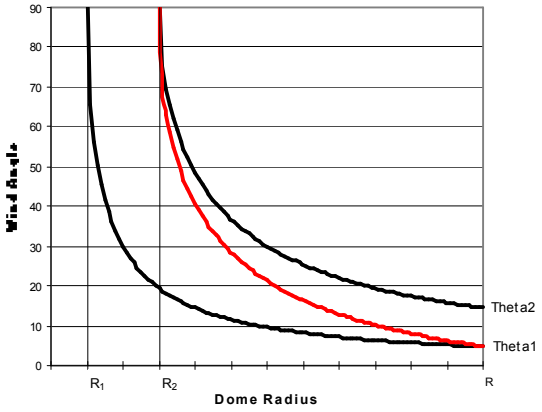


Figure 3.1.3 Wind Angle With-Friction

3.2 Doily Layer

The **DOILY** type layer consists of a washer-shaped ply which is wrapped over other layers as shown in *Figure 3.2*. A doily is constructed with a uniform thickness and usually of an orthotropic material. The primary material direction is rotated from the meridional axis of the tank by the value of the wind angle. For example, a wind angle of 90.0 degrees results in the primary material direction of the doily being in the tank's hoop direction.

The doily geometry is defined by giving an inner and outer radius. The plug-in determines which layers it overlaps and generates any voids if it overlaps the end of another layer.

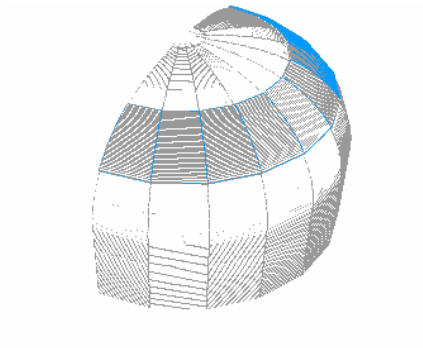


Figure 3.2: Doily Layer in Blue

3.3 Hoop Layer

The **HOOP** type of layer is a uniform-thickness layer which begins at the base of the dome, or in case of an attached cylinder, the base of the cylinder, and terminates at a position along the tank axis as shown in *Figure 3.3.* The termination position is defined relative to the interface between the dome and the cylinder. Thus, if a cylinder is attached to the dome then a negative termination position would define a hoop terminating on the cylinder. Conversely, a positive termination position would define a hoop terminating on the dome.

A hoop layer is usually orthotropic like a doily with the primary material direction rotated from the meridional axis of the tank by the value of the wind angle.

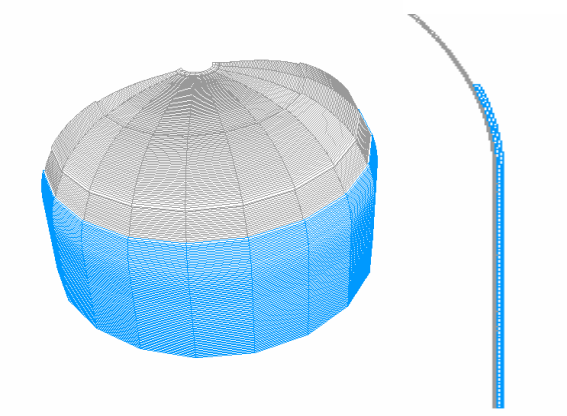


Figure 3.3: Hoop Layer in Blue

3.4 Void Creation

When one layer overlaps the end of another layer, a *single void* will be created as shown in the figure below. The length of the void is equal to the thickness at the termination of the layer being overlapped times the void fraction of the layer being overlapped. The void fraction is a layer-level control parameter which may be modified to contract or expand the size of the void and is shown in *Figure 3.4-1*.

The mesh of the void will always be constructed of triangle elements, or in the case of 3D continuum geometry, wedge elements. The *Wound Composite Dome* dialog requires a resin (or void) material to be assigned to any wound composite dome. This material will be applied to all voids in the dome.

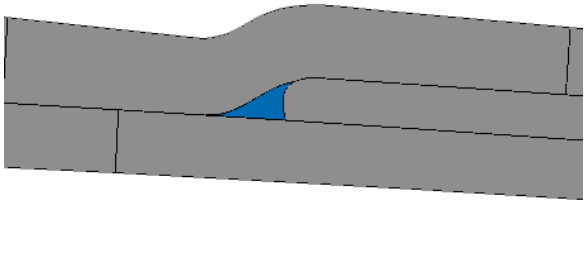


Figure 3.4-1: Single Void

If the outer radius of a doily and the inner radius of another layer are close enough to each other, then a layer which overlaps the two may create what is referred to as a *double void* as shown in *Figure 3.4-2*.

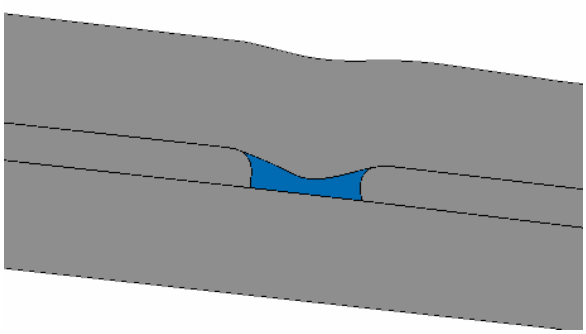


Figure 3.4-2: Double Void

It's possible to wrap a layer over more than two underlying layers which terminate near one another, but the geometry creation in such a case is likely to fail.

The end type of a layer can be assigned as **NO_VOID** in which case no void is created when one layer over wraps another. The shape of the end cap is created such that the slopes at the beginning and end of the end cap generate no discontinuities. This ensures that the layers which are placed over top will in turn not contain any discontinuities.

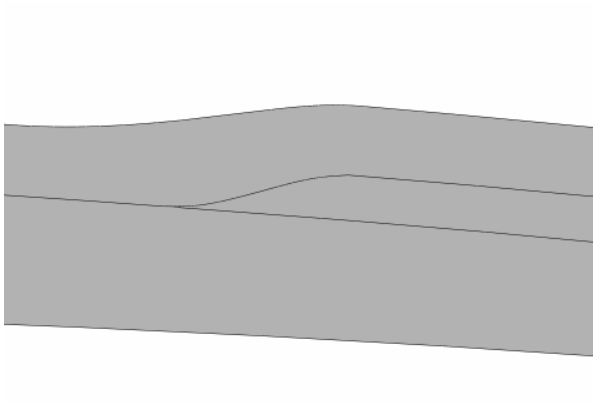


Figure 3.4-3: No-Void End Type

3.5 Layer Termination Options

When on layer terminates near the end of another, it becomes very difficult to determine the geometry of additional layers that may overlap them. For this reason, the default behavior of the plug-in is to disallow one layer from terminating on the end cap of another. The end cap definitions are discussed in detail in *Section 3.6.1*. An option is, however, provided to override this default behavior. The *Transition Point* option will automatically move the turnaround radius of any layer terminating on the end cap of an underlying layer to the radius of the transition point of the underlying layer's end cap. The transition point is simply the beginning point of the end cap.

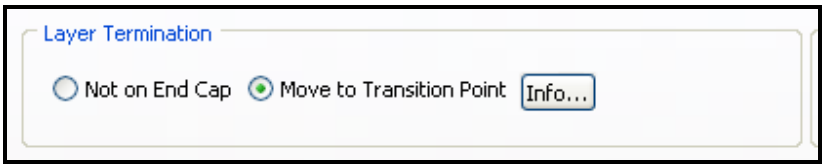


Figure 3.5-1: Layer Termination Options in Wound Composite Dialog

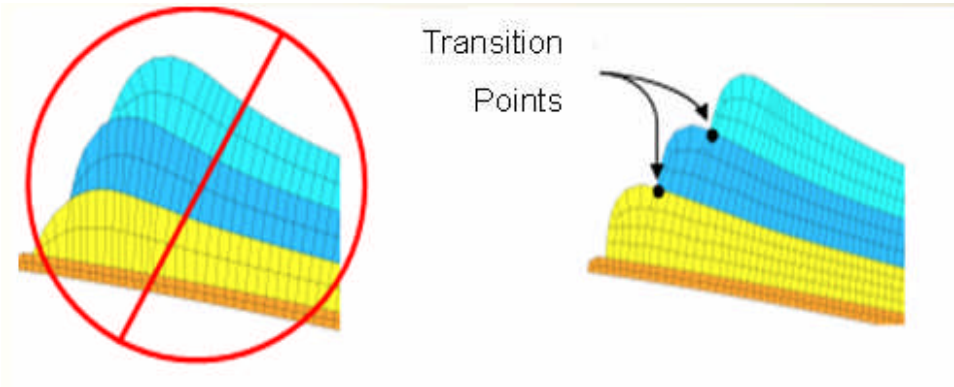


Figure 3.5-2: Layer Termination Options

3.6 Layer-Level Controls

Controls at the layer level are available for each layer type. The controls can be grouped in three categories: end controls, mesh controls, and CAD data fitting controls. The CAD data controls are used for wind angle and band width curve fitting and are only applicable to helicals or hoops with friction. If the **Parameters above** button is selected, then the shape of the layer is determined from the attributes provided and any CAD data is ignored. If the **Least-Squares frit of Cad Data** button is selected, then a least-squares fit is used to determine the curve-exponent and the band width factor. A least-squares fit is not yet available for the extrapolation fraction so a default of 0.5 is used. Therefore, the parameter-based curve should fit the CAD data throughout the length of the dome, except possibly near the end of the layer.

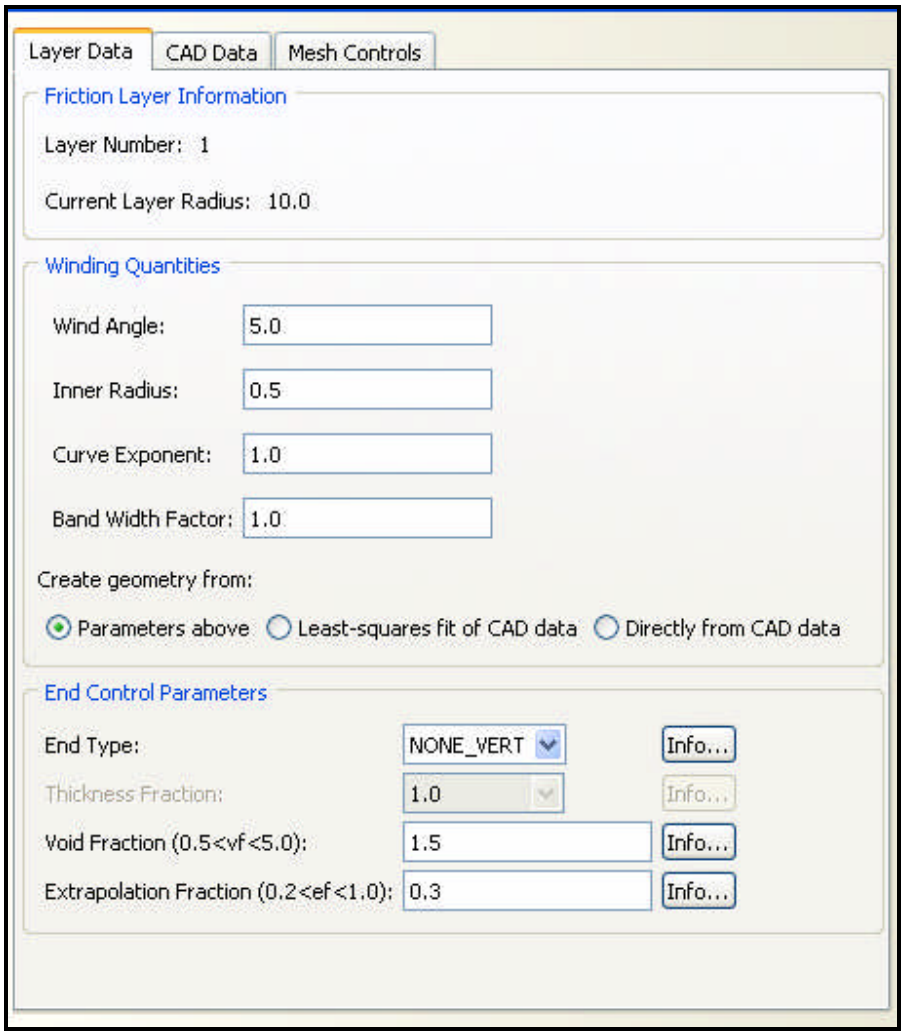


Figure 3.6: Layer-Level Controls

3.6.1 End Controls

Five different end types are available for each type of layer. The **FILLET**, **SPLINE** and **NO_VOID** end types permit the user to specify an end fraction which controls how abruptly the end is rounded off. The **NONE** and **NONE_VERT** end types do not round off the end of the layer.

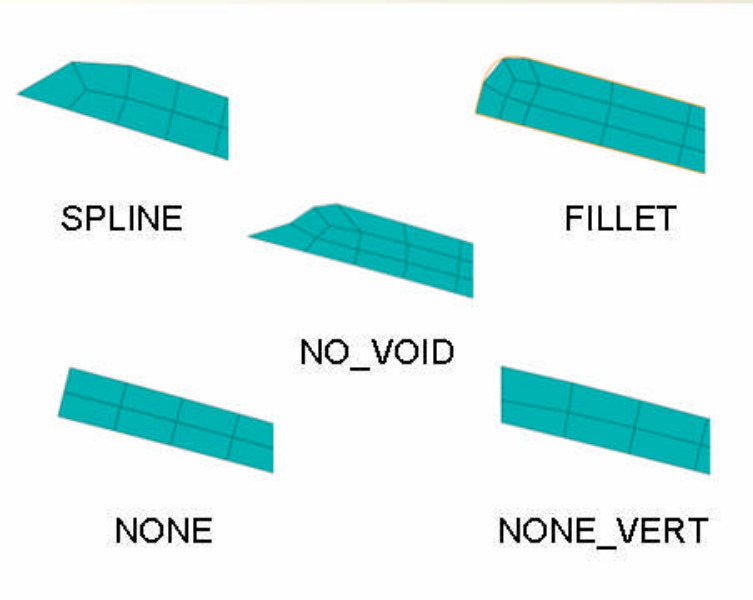


Figure 3.6.1-1: Layer End Types

The **NONE_VERT** end type is used for butting up the layers vertically against a polar boss. Layers with this end type may be stacked upon one another terminating at the same radius as shown in *Figure 3.6.1-2*. This option is used primarily with dome geometries defined from an existing part instance.

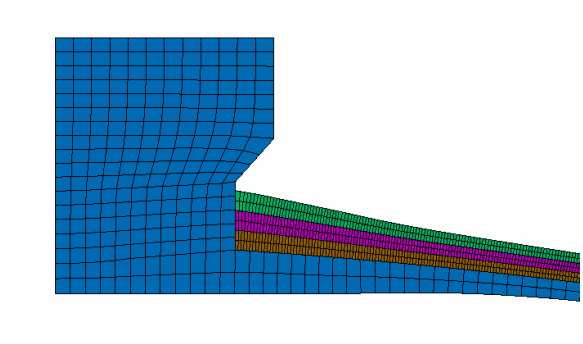


*Figure 3.6.1-2: Layers Abutting Polar Boss via **NONE_VERT** End Type*

An additional end type, **CAD_DATA**, is available only for models which are generated by importing CAD data. The type of these layers is set to either **HELICAL_FRIC** or **HOOP** and is determined automatically upon import. Upon import, the user chooses to either set the end type of each layer either to **CAD_DATA** or **NONE_VERT**. In either the case, the end geometry will be modified depending on whether the CAD data is defined and the center or edge of the band.

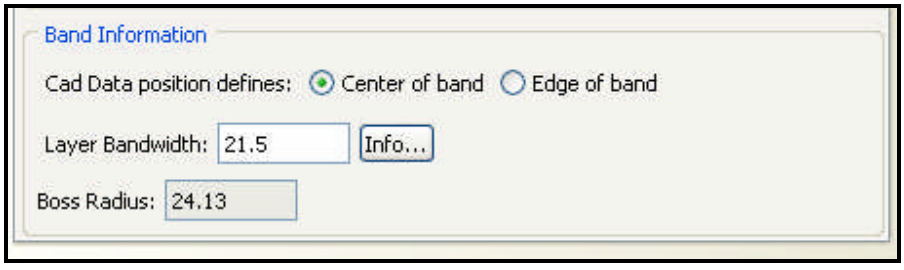


Figure 3.6.1-4: CAD Data Band Width Settings

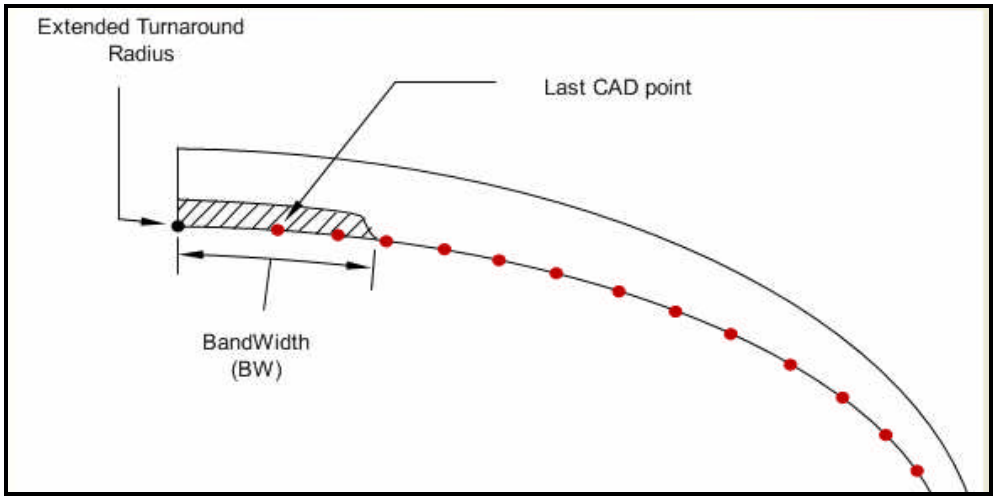


Figure 3.6.1-5: Modification of Turnaround Radius due to Band Width

Setting the end type to **NONE_VERT** will cause the last couple of CAD data points to be modified, if necessary, to ensure the end abuts the polar boss properly and is aligned vertically. The radius of the polar boss is set during the import of the geometry and at present can not be modified. The thickness at the end of the layer, however, may be modified based on the position of the band. If the CAD data positions are measured at the inner most edge of the band, then the thickness at the polar boss will be determined directly from the CAD data. However, if the CAD data positions are measured at the center of the band, then the thickness will be extrapolated from the last CAD data point to a position one half of the width of the band away.

Setting the end type to **CAD_DATA** will instruct the plug-in to use the points defined in the CAD data exactly up to a point which is one half of the width of the band away from the turnaround radius. From this point to the turnaround point the layer thickness in linearly ramped down to zero.

3.6.2 Layer-Level Mesh Controls

The layer-level **Mesh Controls** tab as shown in *Figure 3.6.2-1* allows individual layers to be assigned mesh controls that give the user significant controls over the characteristics and quality of the final mesh. The layer-level controls override the dome-level mesh controls described in *Section 3.7*.

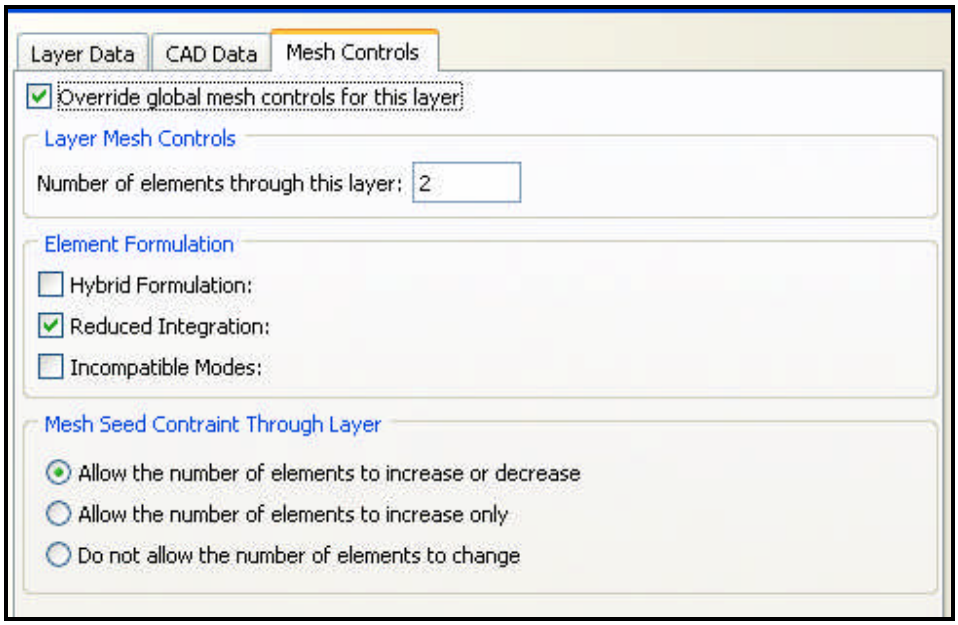

Layer Data
CAD Data
Mesh Controls
Override global mesh controls for this layer
Layer Mesh Controls
Number of elements through this layer: 2
Element Formulation
Hybrid Formulation:
Reduced Integration:
Incompatible Modes:
Mesh Seed Contraint Through Layer
Allow the number of elements to increase or decrease
Allow the number of elements to increase only
Do not allow the number of elements to change

Figure 3.6.2-1: Layer-Level Mesh Controls Tab Input

The number of elements through the thickness of a particular layer may be specified in this dialog. Resulting mesh seeds are applied through the thickness of the layer at the base of the dome, or in the case of an attached cylinder, at the interface between the dome and cylinder. If partitions are prescribed as discussed in *Section 3.7*, the mesh seeds are applied on the segment of the partition passing through the layer of interest. The mesh seeds assigned through the thickness may be constrained to increase, decrease, or remain fixed through this dialog.

The element formulation may be assigned to the layer by selecting any combination of the **HYBRID**, **REDUCED**, or **INCOMPATIBLE MODES** formulations.

3.6.3 CAD Data Fitting

Tables of CAD data lines containing radius, y-position, wind angle and thickness are often available as output from filament winding machines. In order to generate a COPV model from this data, helical layers with friction or hoop layers may be created during the import of CAD data. Selecting the **CAD Data** tab from the layer-level controls dialog shows the CAD data read during the import process. Right clicking on the table provides a method for refilling the table by reading from a text file. The text file must contain four columns of numbers separated by either a comma or a blank space. Individual rows of the table may edited or removed altogether.

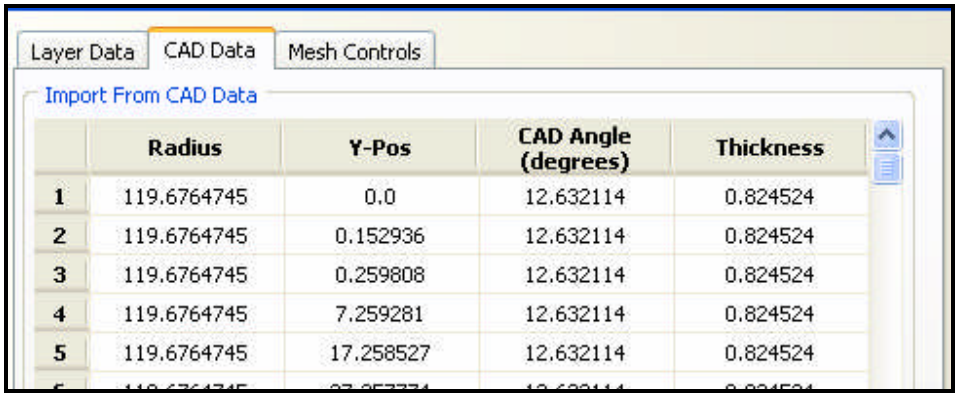


Figure 3.6.3-1: Cad Data Import Tab

The first and last entries of the CAD data define the bounds of the wind angle curve with-friction. The lower bound, as shown in blue in *Figure 3.6.3-2*, is determined from the frictionless wind angle curve of *Equation 2.1* for the wind angle specified from the CAD data at the tangent line. The upper bound is based on the frictionless wind angle curve that would terminate at the turnaround radius specified from the CAD data. The black line shows the default behavior which sets the curve exponent *n* of *Equation 2.1* to 1.0. The green line shows what a curve might look like for a non-default curve exponent of 0.1.

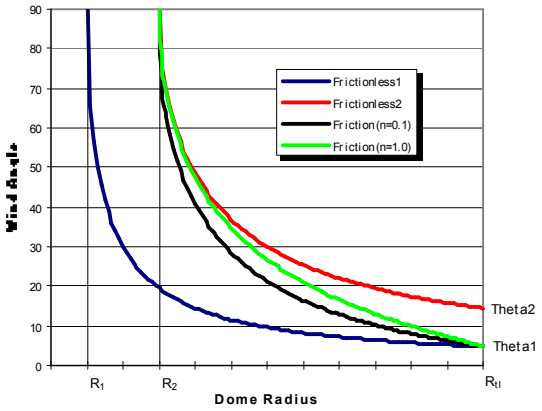


Figure 3.6.3-2

This exponent can be manually entered as shown in *Figure 3.6.3-2*. By selecting the **Least-Squares fit of CAD Data** button, the plug-in uses a least-squares fit to determine the curve exponent *n* and bandwidth factor which best match the CAD data.

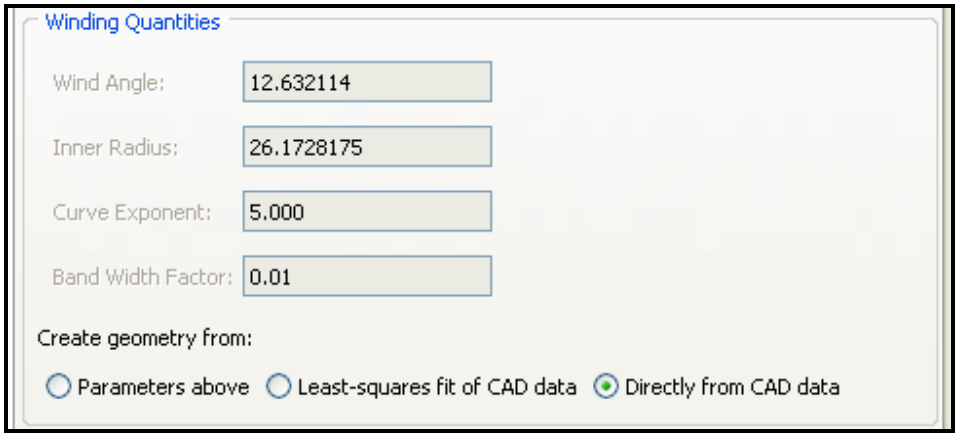


Figure 3.6.3-3: Layer With-Friction Data Controls

A plotting utility has been made available from within the layer controls dialog. This provides a means of immediately determining the quality of the wind angle fit that will be used by the plug-in. The plot can show

the frictionless curve, the curve with friction and the actual curve used for the CAD data fit. An example of a plot is shown in *Figure 3.6.3-4*.

.

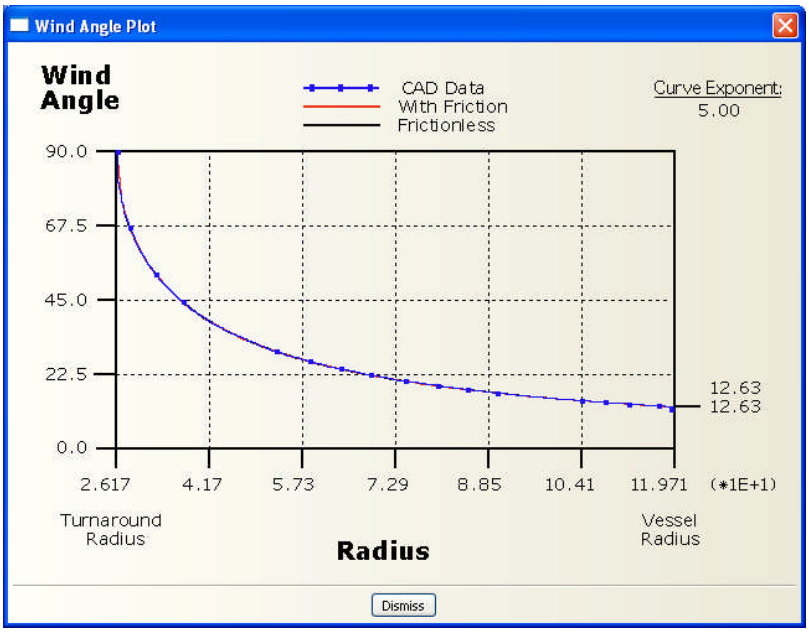


Figure 3.6.3-4: Wind Angle Plot of CAD Data

3.7 Dome-Level Mesh Controls

Dome-level mesh controls are available from the **MESH CONTROLS** tab on the *Wound Composite Dome* dialog. The dome-level mesh controls are applied to all layers of the dome. The number of elements through the thickness, along with the seed constraints which are shown in *Figure 3.7-1,* can be overwritten at the layer-level.

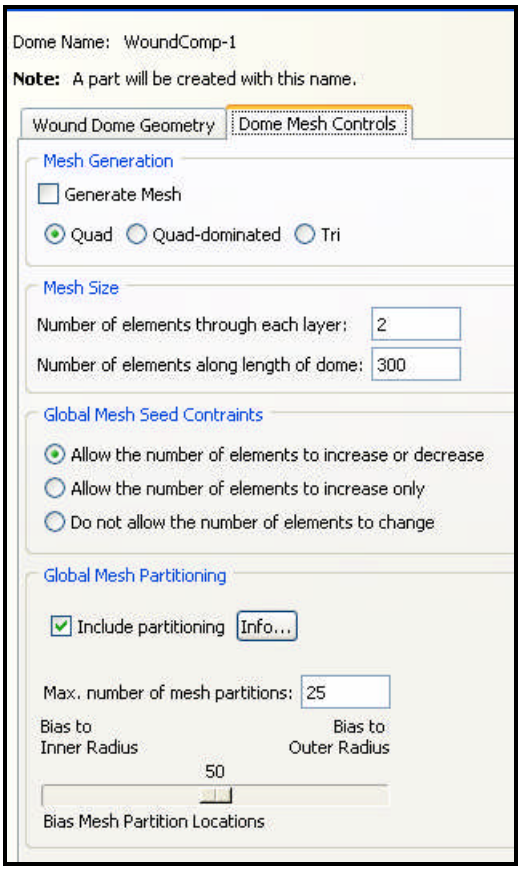

Dome Name: WoundComp-1
Note: A part will be created with this name.
Wound Dome Geometry
Dome Mesh Controls
Mesh Generation
Generate Mesh
Quad
Quad-dominated
Tri
Mesh Size
Number of elements through each layer: 2
Number of elements along length of dome: 300
Global Mesh Seed Contraints
Allow the number of elements to increase or decrease
Allow the number of elements to increase only
Do not allow the number of elements to change
Global Mesh Partitioning
Include partitioning
Info...
Max. number of mesh partitions: 25
Bias to Inner Radius
Bias to Outer Radius
50
Bias Mesh Partition Locations

Figure 3.7-1 Wound Composite Dome-Level Mesh Controls

The mesh partitioning options allow for the creation of partitions through the thickness of a layer all along the length of the layer. The spacing of the partitions can be biased toward the top of the layer, the bottom of the layer, or evenly spaced throughout the layer. The partitions control the mesh in a couple of ways.

Firstly, they prevent the mesher from creating skewed elements as shown to the left in *Figure 3.7-2*. A mesh modified with partitions is shown to the right in *Figure 3.7-2*.

Secondly, they allow for the application of mesh seeds and constraints through the thickness all along the length of the layer as opposed to having only a single mesh seed constraint at the bottom of the dome. This helps to ensure the number of elements through the thickness remains uniform throughout the layer.

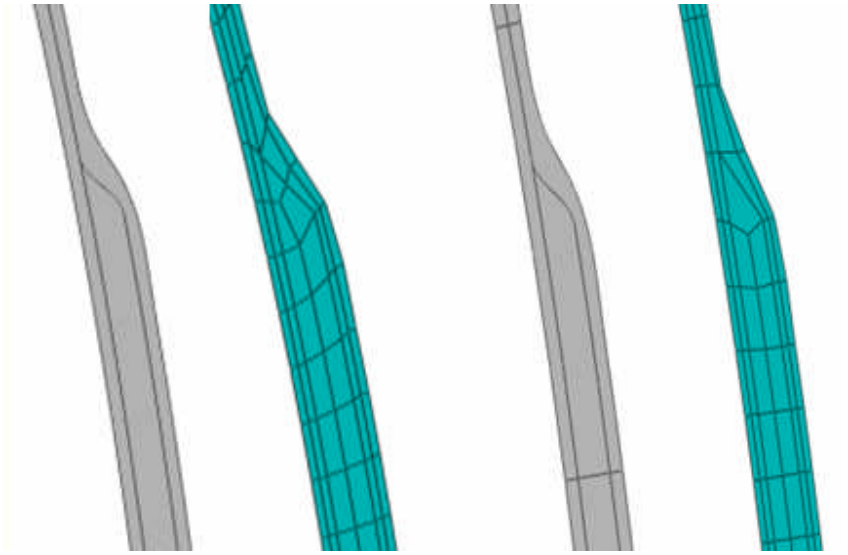


Figure 3.7-2: Geometry/Mesh with and without Partitioning

4. Wound Composite Tank Creation

4.1 Overview

The final creation of the composite overwrapped pressure vessel model is performed by choosing from the library of domes for the top and bottom domes of the tank and selecting the modeling space of the tank via the *Wound Composite Manager* dialog shown in *Figure 4.1-1*. The parts created when generating the domes are axisymmetric and act as a guide when generating tanks of the various available geometries. However, the tank geometry may be axisymmetric or three dimensional

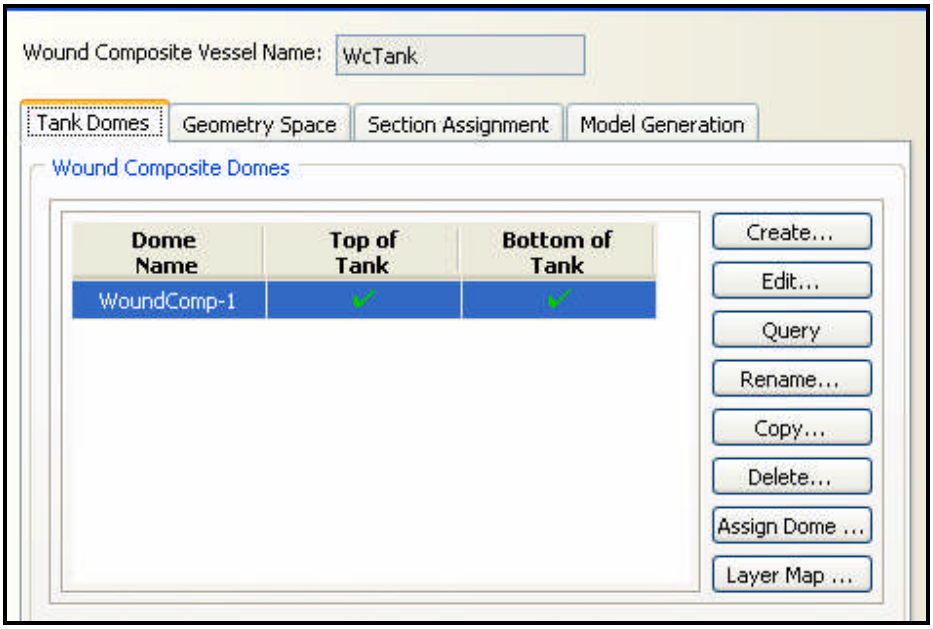


Figure 4.1-1: Wound Composite Tank Manager Dialog

If a top and bottom dome are assigned to the tank, they do not necessarily have to be identical. They are required, however, to have matching layers at the interface between the two. That is to say, the layers at the interface must be of the same type, must have the same thickness, and for helical layers must have the same wind angle. The wind angle may differ between the domes only if one or both domes have cylinders attached. In such a case, the wind angle between the two domes is linearly interpolated over the length of the cylinder.

If either of the domes contains doilies, then the global layer numbers of the full tank will differ from that of the individual domes. All warning and error messages written by the plug-in refer to global layer numbers, so a map between the local dome layers and the global tank layers is necessary. Clicking the **Layer Map** button brings up a dialog providing the map as shown in *Figure 4.1.-2*. In the example shown, the second layer of each dome is a doily.

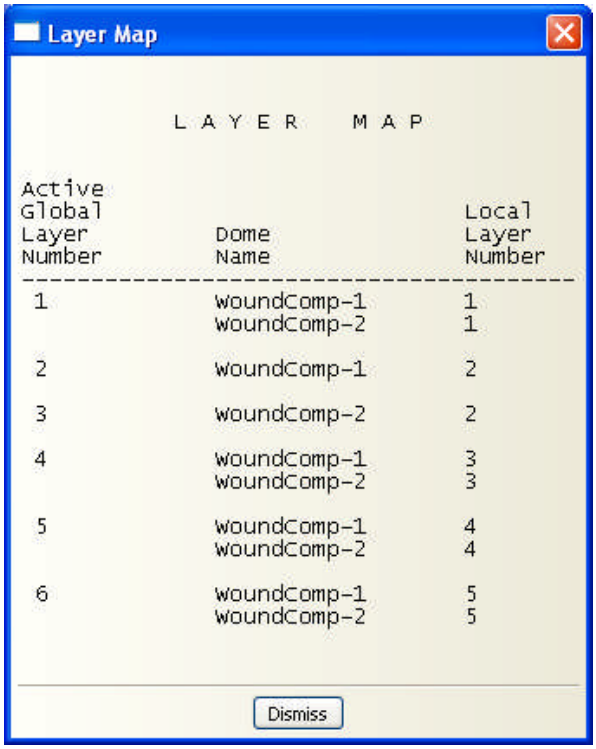


Figure 4.1-2: Layer Map Dialog

4.1.1 Axisymmetric Continuum Geometry

Choosing axisymmetric continuum elements will result in a model exactly the same as the combination of the two axisymmetric domes.

4.1.2 3D Continuum Geometry

If three-dimensional continuum geometry is chosen, a table appears in the Tank Manager Dialog as shown in *Figure 4.1.2-1 below*. The domes chosen provide a two-dimensional sketch which can be swept between 0 and 360 degrees to generate a three-dimensional part. The swept geometry can be broken into equally spaced segments by assigning a number to the *Number of Segments* entry. Partitions are generated in the swept geometry to cut it into equally spaced segments. Each segment may be assigned a different number of elements as shown.

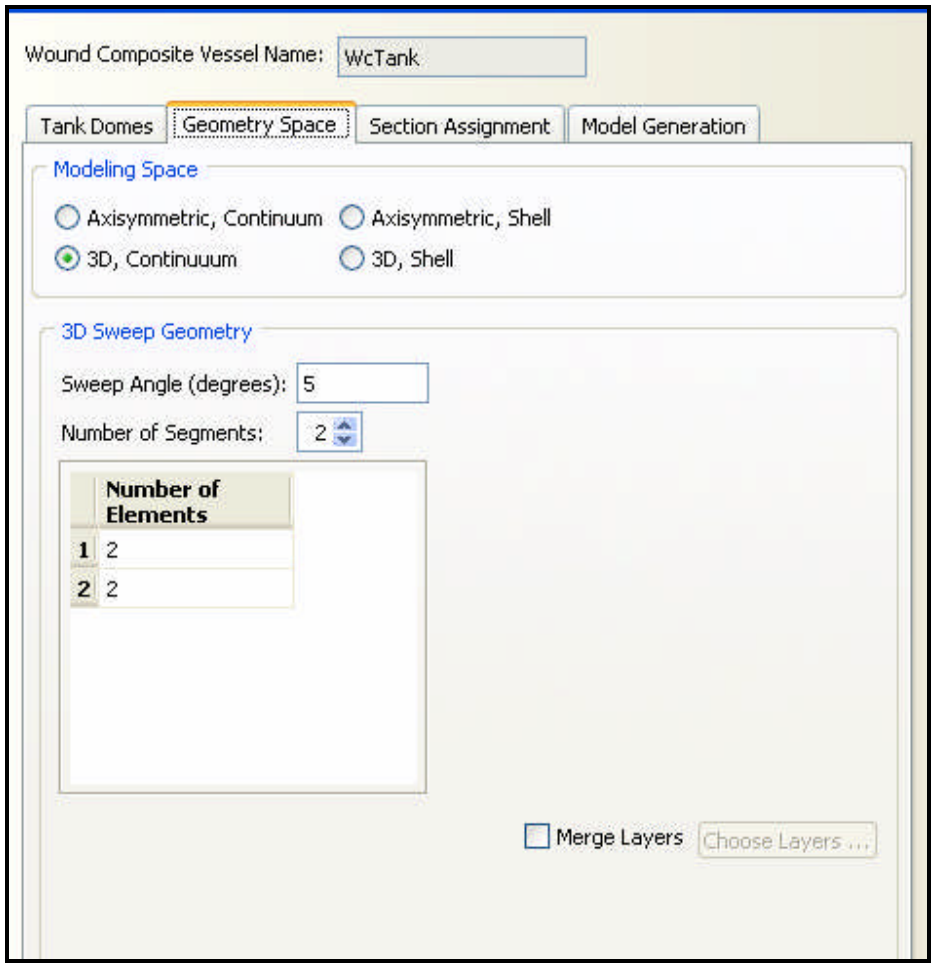


Figure 4.1.2-1: Wound Composite Manager Dialog

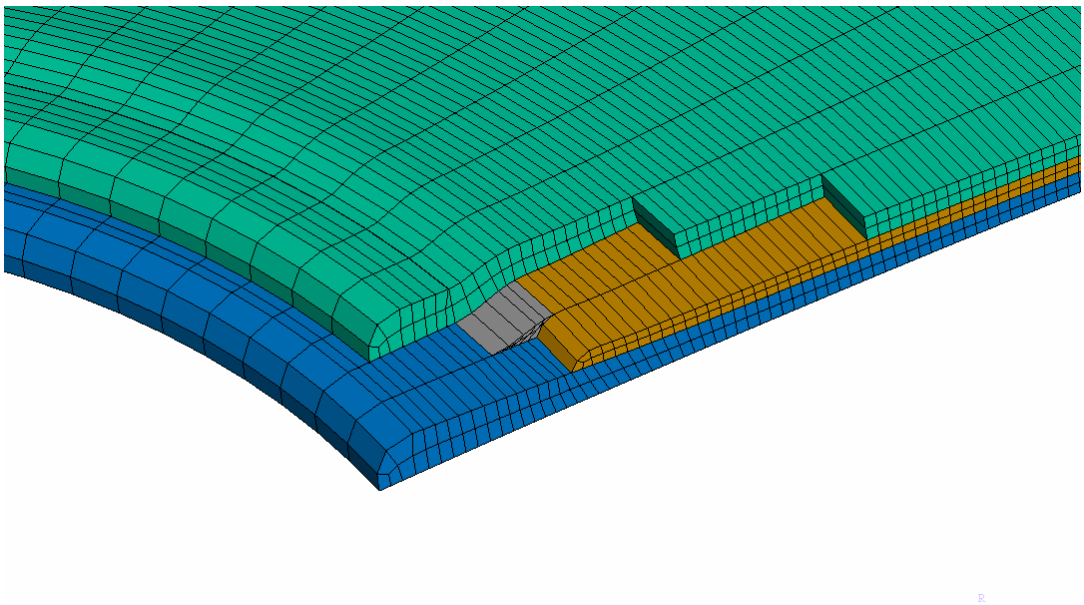


Figure 4.1.2-2: Three-Dimensional Continuum Mesh

4.1.3 3D Shell Geometry

Selecting the **3D, Shell** geometry button will cause the additional options to be activated as shown in *Figure 4.1.3-1.* The number of elements specified is the number of elements along each individual dome, including the cylinder length attached to the dome. A biasing factor is applied to bias the elements toward the polar boss. The larger the biasing number, the more the elements are concentrated toward the polar boss.

The Section Integration options allows the laminated shell section to be integrated either at every iteration throughout the analysis, or once at the beginning of the analysis. Integration only once is intended for cases in which the geometry and material properties of cross-section of the elements do not change significantly during the analysis.

Three-dimensional shell geometry, like the three-dimensional continuum geometry, can be swept between 0 and 360 degrees. Each segment may be assigned a different number of elements as shown in *Figure 4.1.3-2*.

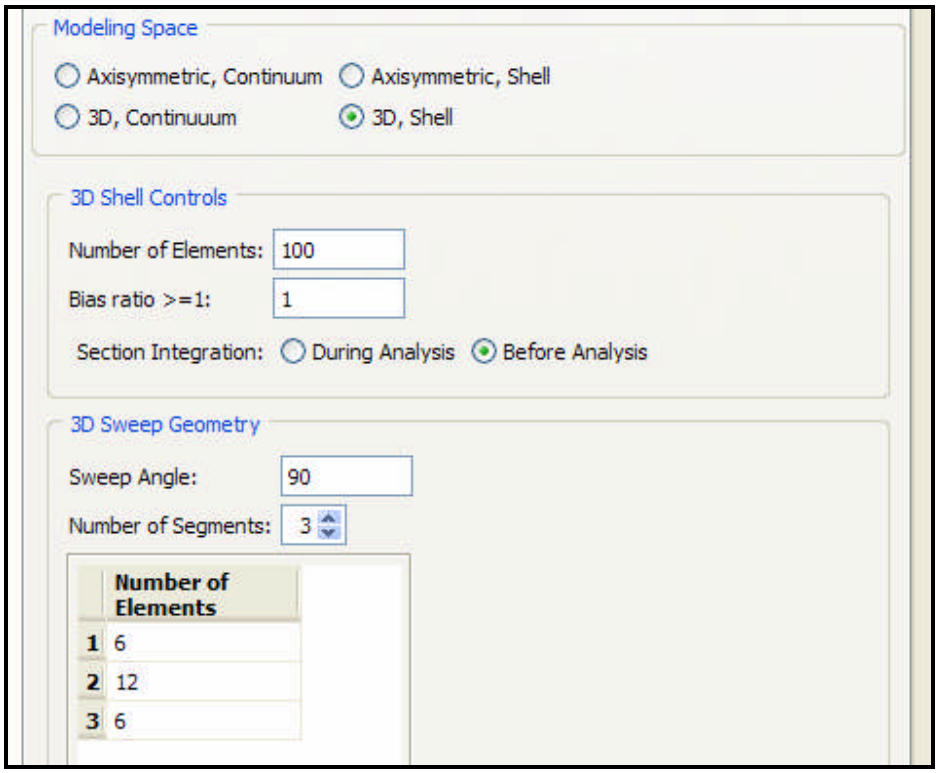


Figure 4.1.3-1: 3D Shell Tank Creation Options

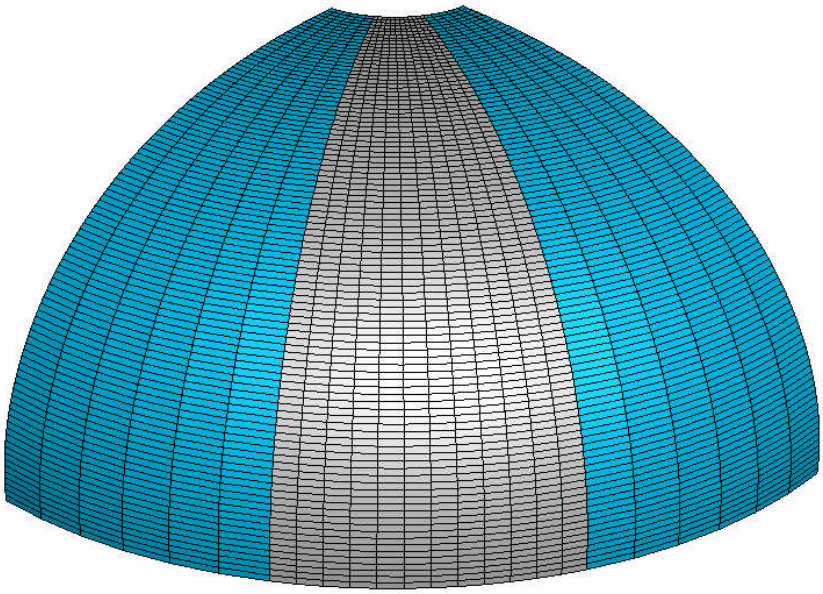


Figure 4.1.3-2: Differing 3D Shell Segment Meshes

The shell section definitions of the elements along are determined by examining the layouts of the assigned domes. The shell element geometry, shown in red in *Figure 4.1.3-3*, is determined from the inner surface of the domes. The shell section properties are specified by calculating the normal at the centroid of each shell element, then determining the stack lay up in the direction of the outward shell normal at the element centroid. The shell section is then offset by half the thickness of the lay up. The thicknesses at the nodes are assigned via the ***NODAL THICKNESS** option by determining the stack lay up thicknesses at the nodal locations. Because of the axisymmetric geometry and wind angle orientation assumptions, the shell section definitions are grouped together in the hoop direction to minimize the number of sections, thus improving performance in the solvers.

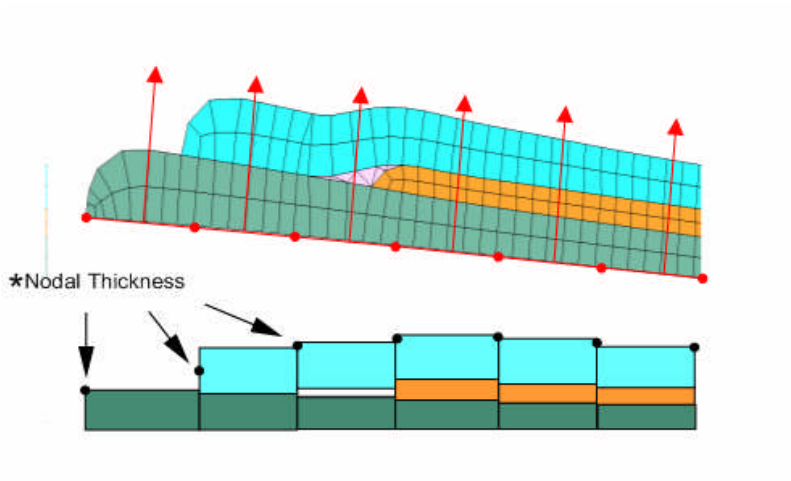


Figure 4.1.3-3: Laminated Shell Sections

4.1.4 Axisymmetric Shell Geometry

Choosing axisymmetric shell geometry results in line geometry created at the inner surface of the axisymmetric domes. The procedure for determining the shell section definitions for axisymmetric shells follows the same procedure as that of the three-dimensional shells. The nodal thicknesses are also calculated in the same manner as the three-dimensional shell elements.

4.1.5 Continuum Shell Geometry

The model size of three-dimensional continuum models can quickly overwhelm the capacities of most hardware systems. In order to reduce the size of three-dimensional continuum models, an option is provided to merge the properties of selected layers into a single laminated continuum shell element. This allows the size of the model to be significantly reduced, while retaining most of the accuracy of the full model.

The process of creating a merged layer continuum shell model follows the same process as that of the standard three-dimension continuum model, except an additional step of selecting the layers to be merged is initiated by selecting the **MERGE LAYERS** button. This brings up the dialog shown below. Layers may be highlighted, then by clicking on the **MERGE SELECTED ROWS** they are assigned the same global layer number shown in the far right column of the dialog. The rows selected during this process must be sequential; .i.e. row 4 can not be merged with row 2 unless row 3 is also merged. Merged layers may consist of a single row. Special care must be taken when merging models with doilies. The exterior of merged layers must result in a single, continuous group of points. For example, an invalid merge definition would result if one doily belonging to the top dome and another to the bottom dome were selected, but no layer joining the two was selected. The result would be the exterior of the merged layer consisting of two unique sets of points.

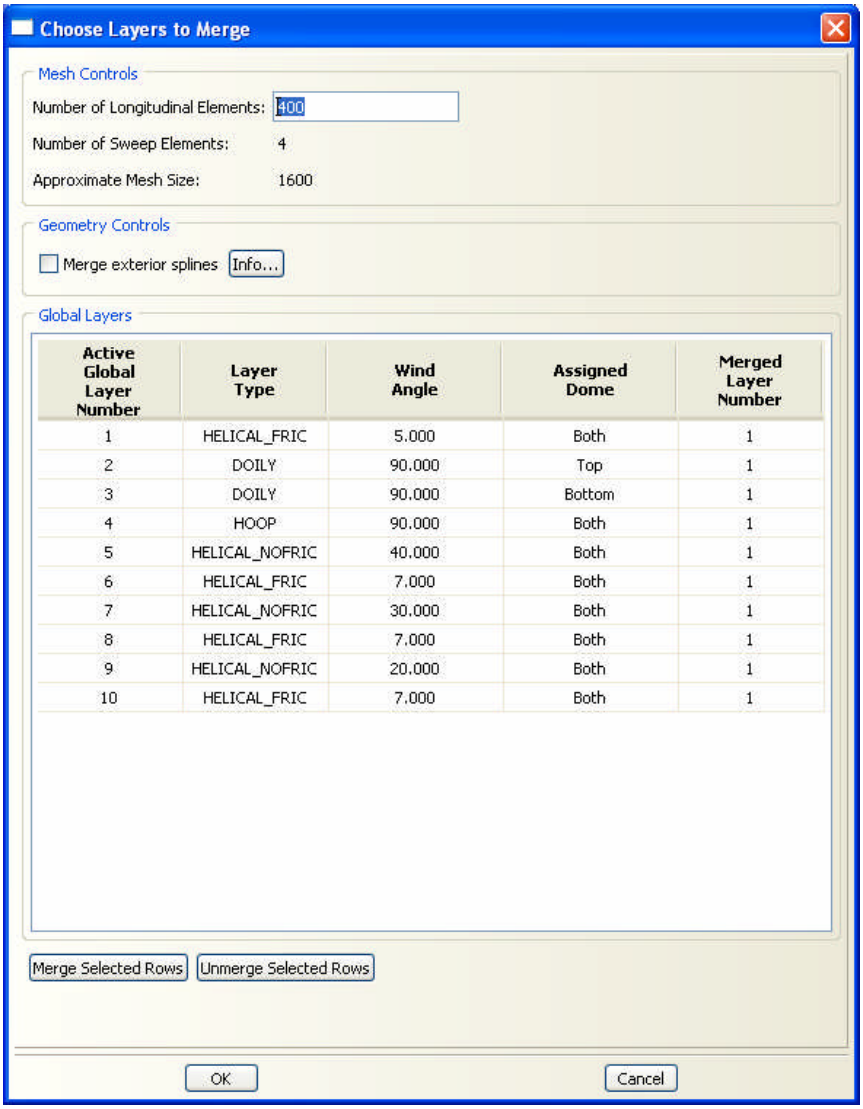


Figure 4.1.5-1: Merge Layer Dialog

Once the layers to be merged are selected, submitting the dialog simply sets the layers to be merged. Not until the tank manager dialog is submitted does the proper geometry get created. The resulting geometry is a three-dimensional continuum shell model in which each layer (consisting of multiple merged layers) is represented by one element through the layer thickness all along the length of the layer. If the mesher puts more than one element through the thickness of any merged layer, then more partitions need to be added to enforce the requirement of one element through the thickness.

The section definition of each element along a merged layer includes a stack containing any merged layers comprising the element. This stack is determined in the same way as that of 3D or axisymmetric shells, except instead of accounting for all of the elements of the tank, only the layers merged within the specific global layer are considered. The figures below show a model before and after the merging of layers.

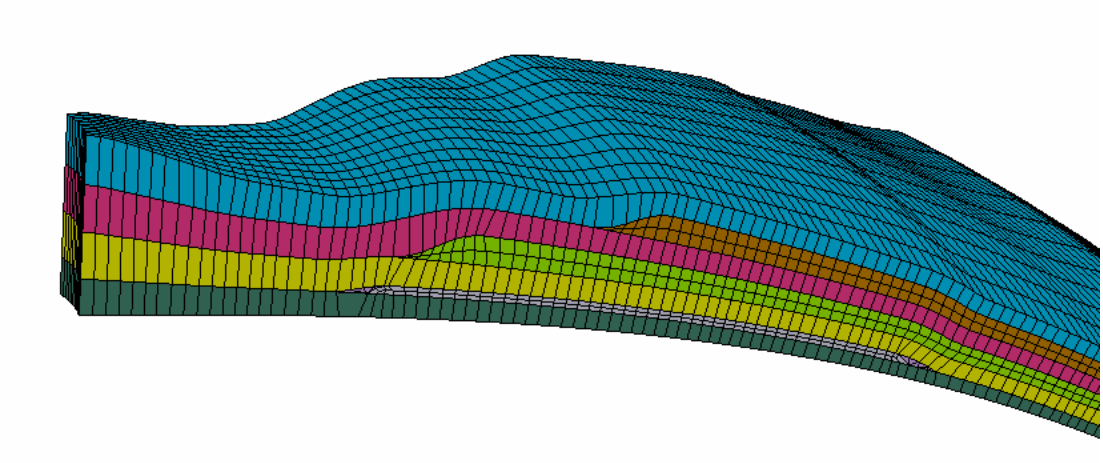


Figure 4.1.5-2: Three-D Continuum Element Mesh

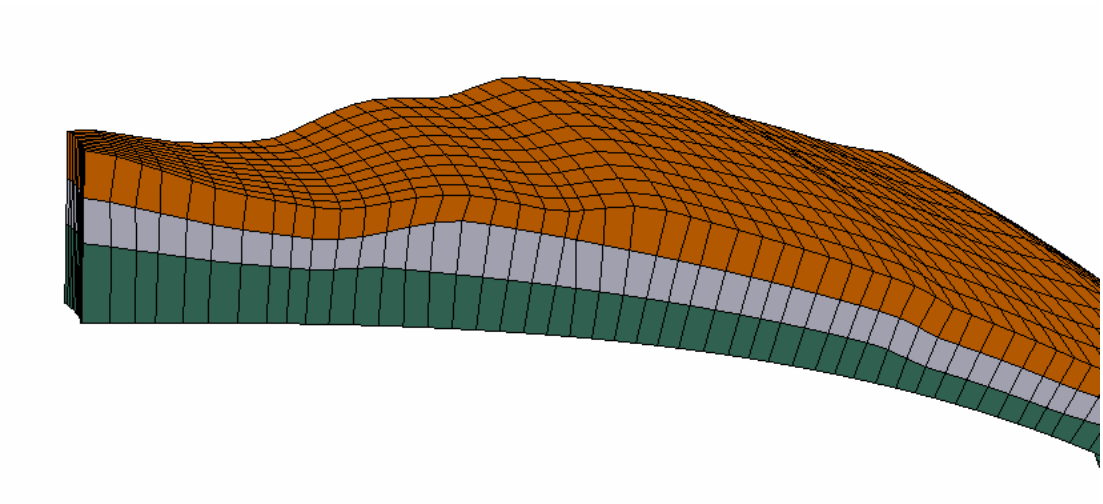


Figure 4.1.5-3: Three-D Continuum Shell Mesh with Merged Layers

When determining the stack of an element, the plug-in searches through the layers along a vector passing through the centroid of the element. The vector is calculated to be normal to the underlying tank geometry (mandral geometry). Since the stack direction of continuum shell elements must be along one of the element's isoparametric directions, the chosen isoparametric direction must coincide as closely as possible with the vector used to determine the stack. For complex geometries which consist of many merged layers generating discontinuous layer geometries, the isoparametric and stack directions can differ significantly. This is because the sketch of the exterior of merged layers is broken into many distinct splines due to the discontinuities. An option has been added to force the exterior of merged layers to be drawn as a single spline. This causes a discrepancy from the actual geometry, however the improvement in the match between stack direction and isoparametric direction more than makes for the discrepancy. Below is shown an extreme case of this phenomenon. Additional mesh partitions can also be added to further ensure that the isoparametric element direction closely matches that of the stack direction (mandral normal).

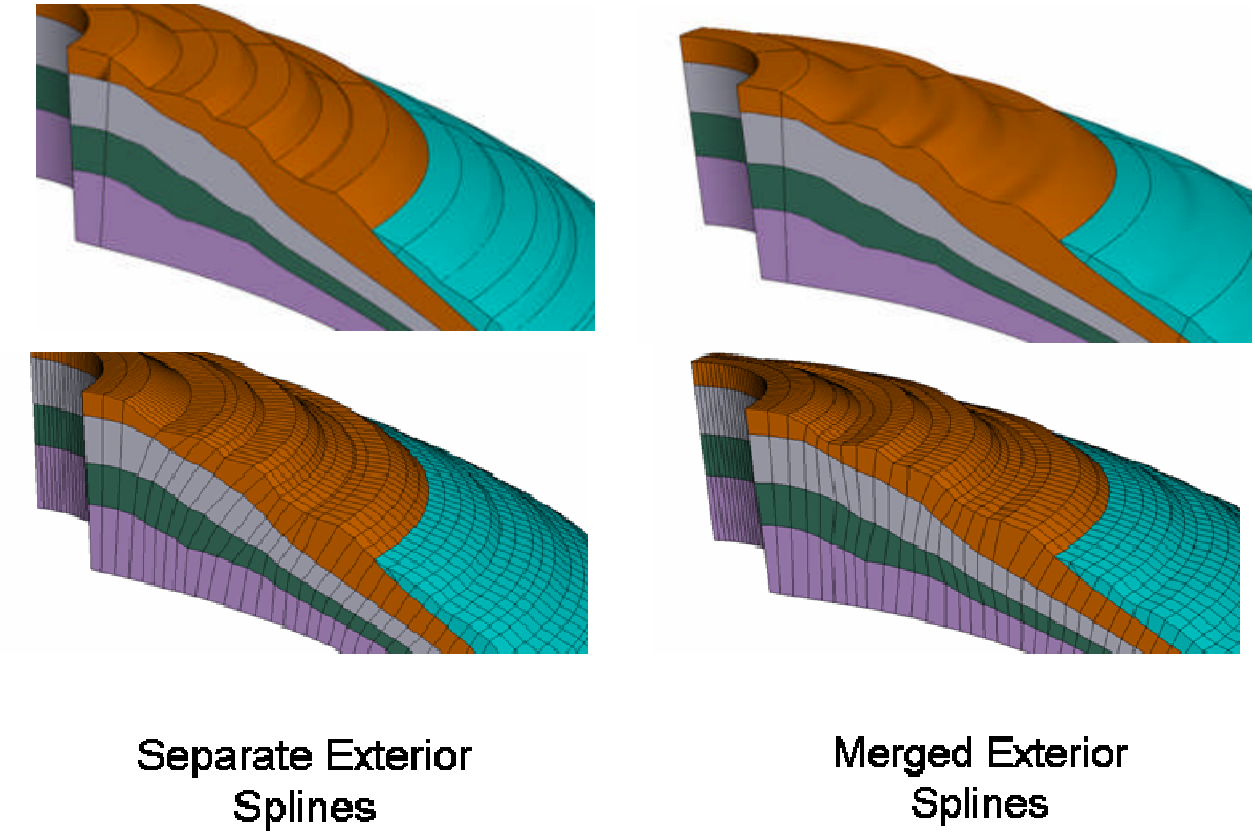


Figure 4.1.5-4: Left: Separate Exterior Splines, Right: Merged Exterior Splines

4.2 Dome Assignments

The assignment of domes is performed by clicking the **Assign Dome** button. The dialog shown in the *Figure 4.2* shows how the top and bottom domes are assigned. A single wound composite dome may be chosen as the top, bottom, or both domes of the tank.

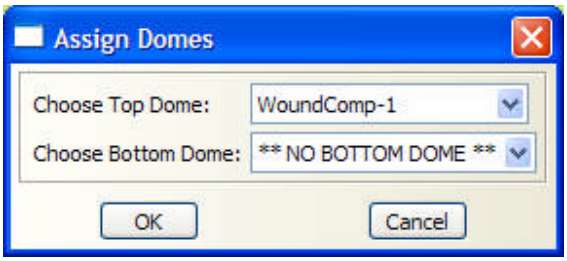


Figure 4.2: Dome Assignment Dialog

4.3 Section Assignments

The options available from the Section Assignment tab determine which element type or types will be assigned to the mesh. Based on the options chosen, the element type(s) are printed at the bottom of the tab area. The groups of the options are arranged in a hierarchy of dependence. At the top of the hierarchy is that of the geometric space defined under the **Geometry Space** tab. All options on the **Section Assignment** tab are dependent on the options chosen above them.

For example, *Figure 4.3* shows the section assignments available for three-dimensional shell elements. The Dynamic Temp-Displacement option under Analysis Type is stippled since Abaqus/Explicit does not provide for the use of three-dimensional shells with this procedure. Both *Linear* and *Quadratic* element orders are available for Coupled Temp-Displacement analyses are available so both buttons can be selected. Since the Quadratic button was selected for the element order, only the *Quad* element shape is un-stippled corresponding to the S8RT element. Thus, only quad-shaped elements are available as second order, three-dimensional shell elements in a Coupled Temp-Displacement analysis. Since there are not hybrid or incompatible Coupled Temp-Displacement shell elements, that option is stippled.

With the exception of element order, section properties follow the same controls scheme as the mesh controls. The element formulation can be assigned globally to all of the layers. Then individual layers may be assigned their own element formulation. The element order is required to be the same throughout, otherwise ***TIE** constraints would be required between regions of first and second order elements. This layer-by-layer control of element type allows the mixture of hybrid elements (for rubber shear-ply materials) with standard or reduced formulation continuum elements for the composite layers.

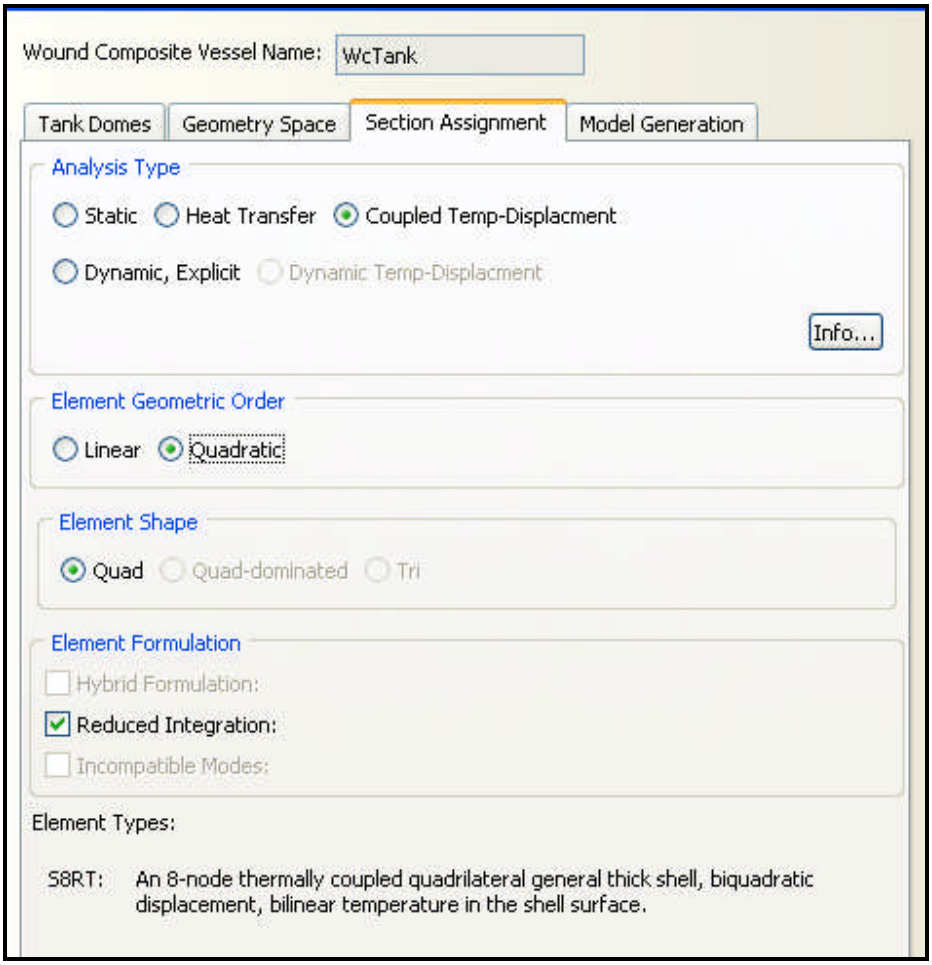


Figure 4.3: Tank-Level Section Assignments (3D Shell)

4.3.1 Material Properties

The assignment of material properties to the tank begins by assigning section definitions based on the materials provided for each layer in the winding layout table. The plug-in then determines which layers have orthotropic materials assigned to them (***ELASTIC**, ***CONDUCTIVITY**, or ***EXPANSION**). The elements in the orthotropic layers are assigned transformed material definitions and section assignments which are written to an include file as described in *Section 4.6.1*.

The calculation of orthotropic materials properties is performed by first creating wind angle bins for each orthotropic material. That is to say, all the elements assigned a given orthotropic material are grouped together based on wind angle and put into bins. The local wind angle of each element is calculated from Equation 2.1 based on the centroidal coordinates of the element. The range of wind angles assigned to a bin is chosen by the user as the wind angle increment in the Tank Manager Dialog as shown in *Figure 4.6*. Wind angle bins are then generated from 0 to 90 degrees based on the wind angle increment. For example, 90 bins would be created for a wind angle increment of 1 degree. All elements with wind angles falling within the range of a given wind angle bin is assigned a single material property based on the wind angle of the bin. No materials are created for bins which have no elements associated with them.

The orthotropic materials for each wind angle bin are calculated as angle-ply laminate (±θ) materials, where the angle θ is the bin wind angle. The orthotropic material properties input by the user are for the composite lamina (single-ply) with 1=fiber direction, 2=transverse and 3=normal. The plug-in transforms these material properties to the global directions: 1=meridional, 2=hoop, and 3=normal. These transformed material properties, along with a section definition, are assigned to each wind angle bin and written to include files described in *Section 4.6.1*. Material properties other than orthotropic ***ELASTIC**, ***CONDUCTIVITY**, and ***EXPANSION** are assigned directly to the wind angle bin material without being altered and are also written to the include file.

4.4 Load and Boundary Assignments

The load options consist of thermal loads, an internal pressure load, and boundary conditions. The thermal load options simply allow a uniform, initial (curing) temperature distribution to be applied in the model definition and a final, uniform (room) temperature to be applied during any subsequent step of the analysis. In analyses where temperature is a degree of freedom, the curing temperature is applied as an initial field quantity, and the room temperature load is applied as a boundary condition. In analyses where temperature is not a degree of freedom, both temperature loads are applied as field quantities.

An internal pressure load is available by entering a value for MEOP (maximum expected operating pressure) shown in *Figure 4.4*. The step in which the pressure load is to be applied is also required. If the dome geometry was defined from an existing part, then the MEOP pressure load will be ignored since the surface that the load would be applied is not known. The MEOP load would, therefore, need to be applied from within the load module of Abaqus/CAE.

Finally, an option to apply boundary conditions on the symmetry planes is made available. For a single dome tank, checking the boundary condition toggle will result in the creation of a boundary condition in which the entire symmetry edge is constrained in the y-direction. Three-dimensional models swept 180 degrees or less will have boundary conditions applied to the two planes. The boundary conditions will be transformed to the proper cylindrical system. For tanks with a top and bottom dome, the boundary condition at the interface is only applied to a single node to constrain the tank.

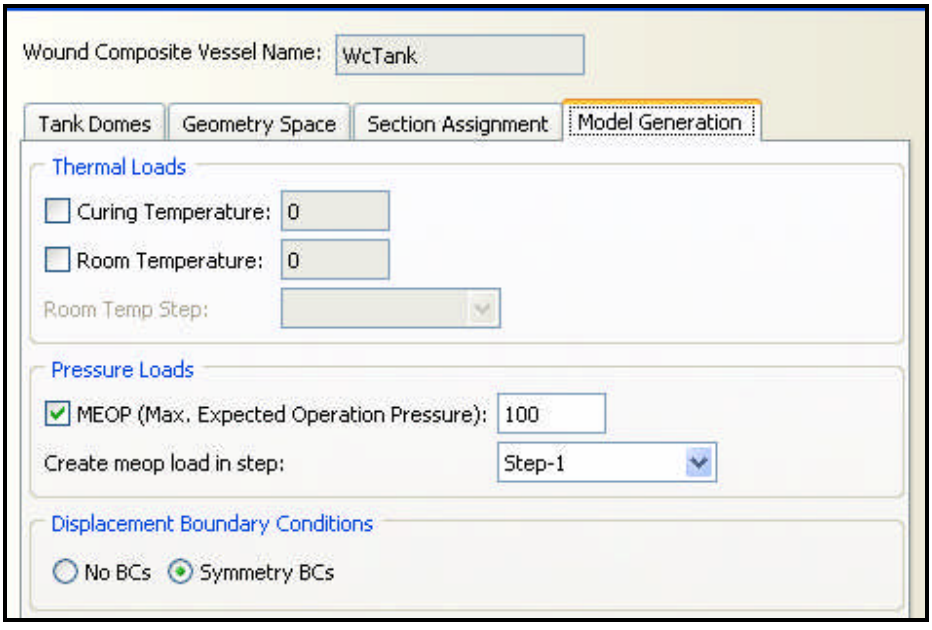


Figure 4.4: Loads Assignment

4.5 Mesh Creation

The next tank information to be specified is the number of elements to be generated along the length of the tank. A toggle is provided to turn off meshing. This is useful when generating large models, thus allowing multiple passes of geometry meshing while skipping the very time consuming step of meshing the part.

The number of elements specified is assigned as the total number assigned to the top and bottom domes. The number assigned to each is specified to generate equal length elements along both domes.

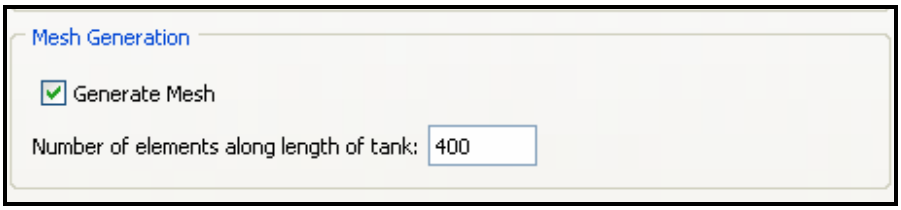


Figure 4.5: Mesh Generation Inputs

4.6 Job Creation

With the domes assigned, the number of elements specified, and the loads chosen, all that remains is to choose to generate the material properties and output if desired. *Figure 4.6* shows the options associated with these under the **MODEL GENERATION** tab.

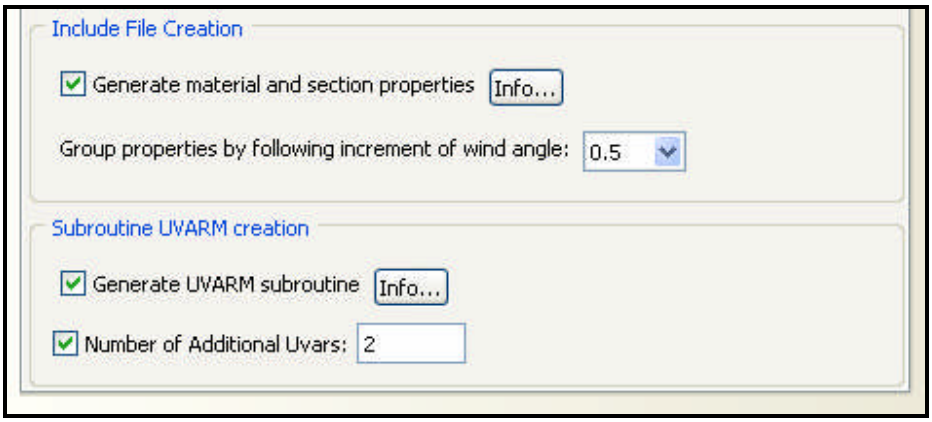


Figure 4.6: Model Generation Inputs

4.6.1 Include Files

Turning on the toggle for generating the material and section properties will transform the material properties which are defined along the fiber direction to material properties defined in the global axisymmetric coordinate system. Since the orientation of the fibers is different at each element of a helical layer, a separate material and section assignment would be required as described in *Section 4.3*. Creating this many material definitions and section assignments within Abaqus/CAE would make editing them almost impossible and would slow down the CAE interface significantly. Thus the properties are grouped into bins. So the plug-in assigns a single material and section definition to each layer from within Abaqus/CAE, then overwrites them by inserting two ***INCLUDE** statements into the keyword editor. One ***INCLUDE** statement referencing

TankName_sections.inc is inserted into the part block. It contains all of the section definitions. The second ***INCLUDE** statement referencing TankName_materials.inc is inserted into the model definition. It contains the material definitions and material orientation definitions.

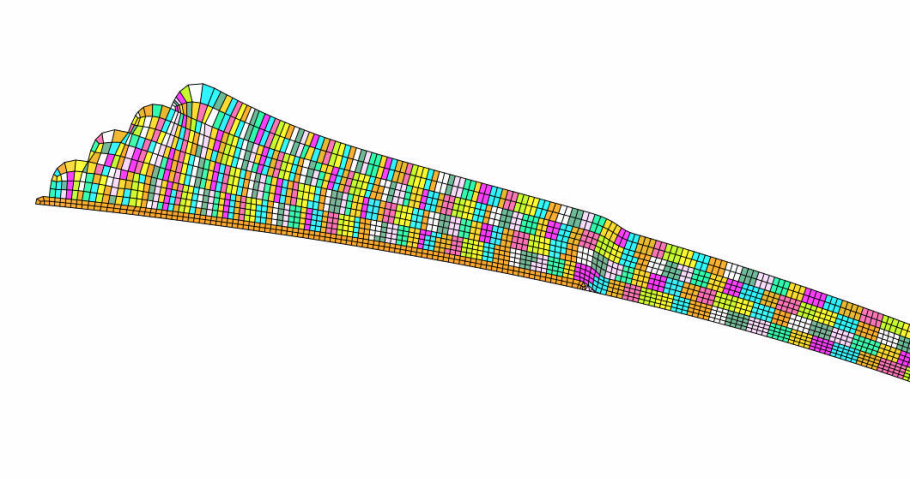


Figure 4.6.1: Wind Angle Bins of 0.5 Degrees

When the job is run, the batch pre-processor will print a warning that numerous elements have redundant section properties and that the last section property will be accepted. This is the expected behavior.

4.6.2 Uvarm Subroutine

Since the material properties of the fiber have been smeared and transformed into axisymmetric coordinate system, output quantities such as stress and strain are not readily available along the fiber direction. For this reason, a toggle has been added to allow the creation of a UVARM subroutine which facilitates the calculation of material properties along and transverse to the fiber direction. An output request is automatically generated requesting Field Output for the corresponding UVARM variables.

By default, the plug-in creates five UVARM output variables for axisymmetric shell and continuum geometries which are the wind angle (UVARM1), the strain along the fiber direction (UVARM2), the strain transverse to the fiber direction (UVARM3), the stress along the fiber direction (UVARM4), the stress transverse to the fiber direction (UVARM5). Because we have the logarithmic strain in the pressure vessel coordinate reference frame and we know the wind angle at each point along the dome, we are able to rotate these strains into a fiber direction coordinate reference frame. For three-dimensional analyses, additional terms are added for fibers at the negative of the wind angles. Specifically, the output variables are the wind angle (UVARM1), the strain along the fiber direction in the positive wind angle direction (UVARM2), the strain along the fiber direction in the negative wind angle direction (UVARM3), the strain transverse to the fiber direction in the positive wind angle direction (UVARM4), the strain transverse to the fiber direction in the negative wind angle direction (UVARM5), the stress along the fiber direction in the positive wind angle direction (UVARM6), the stress along the fiber direction in the negative wind angle direction (UVARM7), the stress transverse to the fiber direction in the positive wind angle direction (UVARM8), and the stress transverse to the fiber direction in the negative wind angle direction (UVARM9).

For Heat Transfer analyses, UVARM2 and UVARM3 are filled with the heat flux along and fibers and transverse to the fibers. For Coupled Temp-Displacement analyses, UVARM10 and UVARM11 are added as the heat flux along the fiber direction and transverse to the fiber direction.

An option is provided for allocating more memory for user-output variables. This is useful if the UVARM subroutine is to be expanded to include more user-defined output variables. The plug-in then sets the ***USER OUTPUT VARIABLES** keyword option in every material definition automatically. A source file, wcUvarmUtils.py, is provided to allow automatic merging of user-defined UVARM coding with that created by the plug-in. Two functions are provided: *writeDeclarations* and *addExtra*. The first inserts declarations statements immediately

following the generic declaration statements of the UVARM subroutine described in the Abaqus User's Manual. The second inserts coding immediately following the plug-in's coding for filling its default UVARM variables. The *uvarm* argument passed into both routines is the file object to be written to. The *nextUvarm* argument is the first available UVARM (an integer) for user-definition. This number is typically is 4 for heat transfer, 8 for coupled temp-displacement, and 6 for all other procedures. The UVARM subroutine is not available in Abaqus/Explicit so the UVARM toggle is stippled for the procedures related to Abaqus/Explicit.

```
def writeDeclarations(uvarm):
    uvarm.write(" C    User-Defined Declaration Statements  \n")
    uvarm.write("      REAL pi\n")
    . . . .

def writeExtra(uvarm, nextUvarm):
    uvarm.write("C      User-Defined Uvarm Coding \n")
    uvarm.write("       CALL GETVRM('PE',ARRAY,JARRAY,FLGRAY,JRCD,JMAC, ,\n")
    uvarm.write("     & JMATYP,MATLAYO ,LACCFLA)    \n")
    . . . .
```

Figure 4.6-2: User-Defined Modifications to UVARM Subroutine

4.7 Externally-Made Changes to Model

Submission of the *Wound Composite Tank Manager* dialog generates a complete, run-ready model of a COPV. However, if changes need to be made to the part representing the COPV, then the following guidelines should be followed for the COPV part:

- No changes to geometry should be made
- Mesh partitions can be added
- Mesh seed changes and remeshing is permitted
- Material changes are permitted

Changes to non-COPV parts are permitted and they may be tied or may interact with the COPV part as desired.

Once any changes to the COPV part have been made, then the material properties need to be recalculated. The **REGENERATE PROPERTIES** button of the *Wound Composite Tank Manager* dialog brings up the dialog shown in *Figure 4.7* Submission of this dialog will cause the material, section, and orientation properties to be recalculated and the include files will be updated accordingly.

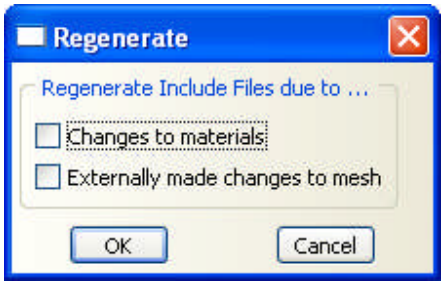


Figure 4.7: Include File Regeneration Dialog

5. Model Creation from CAD Import

The creation of a model can be performed automatically by importing CAD data which defines each layer on a point-by-point basis. The underlying dome geometries are defined via a single mandral file. The geometry of each layer is defined by a ply file for each layer. The mandral file (.man) is formatted as below with the first number defining the y-position along the tank and the second number defining the inner tank diameter at that position.

Horizontal_Position_1, Diameter_1

Horizontal_Position_2, Diameter_2

. . .

The layer files are named with a special convention described below to allow the plug-in to automatically read all the layers at once. The .ply files must all be located in the same directory.

***Name*_ply*n*_*m*.ply**

Where ***Name*** = arbitrary string, same for all layers

n = ply number starting with 1

m = arbitrary string (often wind angle)

The ply data are written in the following format:

Horizontal_Position_1, Diameter_1, Wind_Angle_1 (degrees), Thickness_1

Horizontal_Position_2, Diameter_2, Wind_Angle_2 (degrees), Thickness_2

. . .

The diameter is defined as the diameter at the interior of the layer. The exterior of the layer is determined by first generating a vector which passes through the point and is normal to the mandral. Using the thickness and this normal, a point on the exterior of the layer is calculated.

5.1 Sketch of CAD Data

The first step in creating a model from CAD data is to first read in the CAD data and convert the raw CAD data into a sketch in order to diagnose any potential problem areas. This done by selecting the **Plug-Ins->WoundComposites->Import CAD Data** item under the plug-in drop-down menu. Select the **Sketch Raw CAD Data** toggle and submit the dialog. Another dialog will appear in order to select the mandral and layer files as described above. Creating a sketch with straight lines between every CAD data point would be a very slow process because the constrained sketcher would create a constraint at every data point. An alternative is to draw the data points with a spline. In this case the constrained sketcher only creates constraints at the beginning and end of the spline. The intermediate points are used in the sketch, but are not visible when viewing or editing the sketch. The result is a sketch which is much faster to generate. However, any discontinuities in the normals of the surface will be smoothed out, thus the representation of the layers is not exact.

5.2 Import of CAD Data

To generate a model from CAD data, select the **Plug-Ins->WoundComposites->Import CAD Data** item under the plug-in drop-down menu. Then select the **Import CAD Data** toggle and submit the dialog. The resulting dialog that is brought up is shown below in *Figure 5.2*.

The first step is the selection of the mandral file. The **Select…** button allows you to browse your machine to select the mandral file. The mandral and ply files do not have to be in the working directory. Once the mandral file is selected, the plug-in scans through the file and populates the dialog with the top and bottom boss diameters, the maximum tank diameter found, and the y-pos that represents the center of the tank. The top and bottom domes will be separated at this y-pos. This information is used as a quick check on the data.

The next step is to select a single ply file. Any of the ply files can be chosen. The plug-in examines the name of the file selected and based on the naming convention described above scans the directory for any more ply files. Each ply file is scanned and the layout is filled as shown in *Figure 5.2.1-.1.*

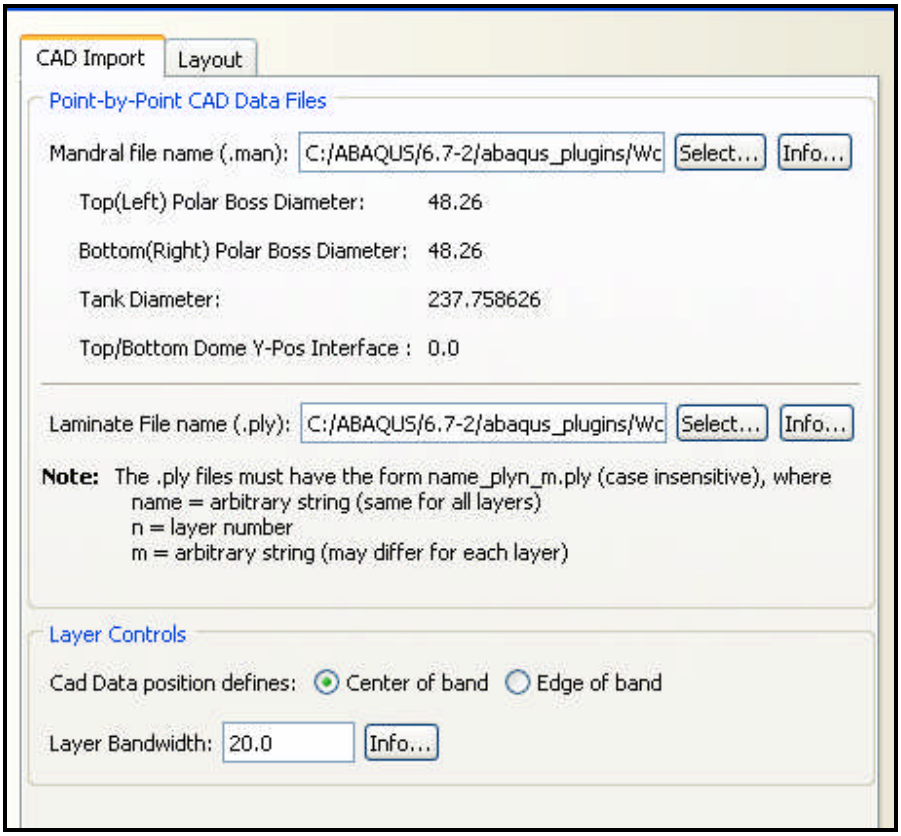


Figure 5.2: CAD Data Import Table

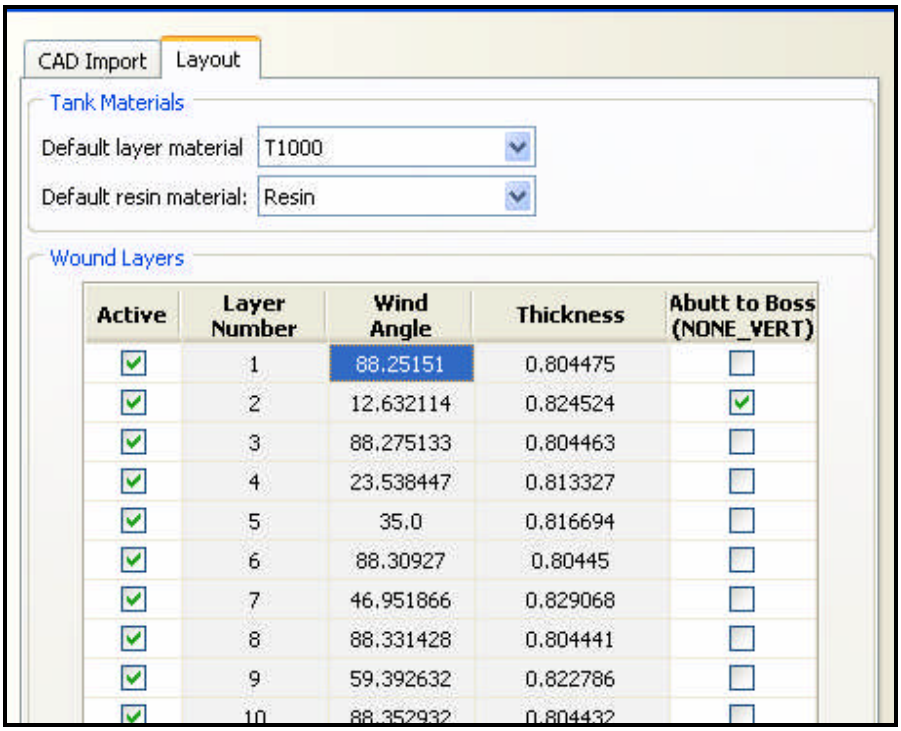
5.2.1 Band Width Settings

When importing CAD data, the band width must be known for a couple of reasons. First, the positional CAD data may be measured from the center of the band or from the innermost edge of the band. If the position is measured from the center of the band, then the inner most point of the layer (turnaround radius) must be adjusted by half of the band width. The plug-in must fill in this missing shape. It does so by tapering the layer thickness down to zero over a length equal to half the band width. This is done for all layers unless they are specified as layers abutting the polar boss as described in the next section.

For high angle helicals (above 85 degrees), the distance over which the end thickness is tapered to zero must be modified. If one half of the width were used to sketch the tapered end region of the layer, the result would be a long, skinny taper which would require a very fine mesh. Instead, the band width of the layer is set to a three times that of the thickness at the end of the layer. This allows for a more reasonable transition and a coarser mesh to be generated around the layer ends. *Figure 5.2.4* shows an example of this. After the initial import of the CAD data, the bandwidth of individual layers can be modified if desired. These high angle helicals are set to the **HOOP** layer type.

5.2.2 Ends Abutting Polar Boss

The CAD data usually does not exactly define the exterior of a layer which abuts the polar boss. For this reason, during the import of the CAD data the user must select which layers will abut the polar boss. The **NONE_VERT** end type will be applied to these layers. All of these layers will terminate at the polar boss radius defined via the mandral file. Even if the adjustment due to band width extends beyond the polar boss radius, the layer will be forced to terminate exactly at the polar boss radius.



Active	Layer Number	Wind Angle	Thickness	Abutt to Boss (NONE_VERT)
☑	1	88.25151	0.804475	☐
☑	2	12.632114	0.824524	☑
☑	3	88.275133	0.804463	☐
☑	4	23.538447	0.813327	☐
☑	5	35.0	0.816694	☐
☑	6	88.30927	0.80445	☐
☑	7	46.951866	0.829068	☐
☑	8	88.331428	0.804441	☐
☑	9	59.392632	0.822786	☐
☑	10	88.352932	0.804432	☐

Figure 5.2.2: END Data Import Table

5.2.3 Tank Creation

Upon submission the CAD Import dialog, the layer data is sorted and assigned to a dome. Checks are made to determine if the tank is symmetric. If so, a single dome is applied to both the top and bottom of the tank. As a default, an axisymmetric model of the entire tank is generated. The CAD data is stored for each layer and may be modified. If using the exact geometry is not practical due to discontinuities in the data, a least-squares fit of the CAD data may be chosen for any given layer and the tank geometry may be regenerated.

5.2.4 Tank Geometry Validation

A comparison between the sketch of the raw CAD data and the sketch used to create the tank can be generated in order to verify that the tank was generated properly. To do this, edit the Wc_Exterior sketch, then insert the *WoundCompName*_LayerPartitions sketch into the existing sketch where *WoundCompName* is the name of the dome used for the top or bottom of the tank. If both domes are being used, then the top and bottom dome sketches will need to be inserted. Finally insert the CAD_PlySketch into the existing sketch and you will see a sketch similar to the one below. Keep in mind that the lines being sketched from the CAD data represent the bottom of the layer while the lines being sketched from the tank model represent the top of the layer. In the sketch below you can also see where the plug-in filled in the unknown geometry of the tapering ends.

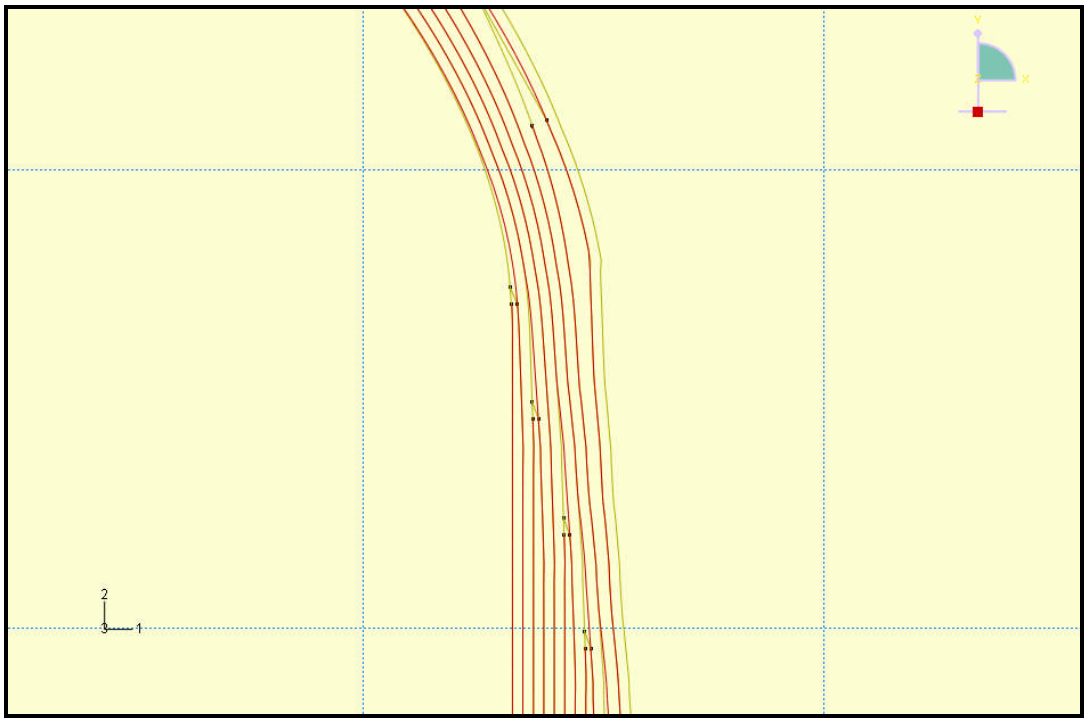


Figure 5.2.4: Raw CAD Data vs Tank Sketch

5.2.5 Diagnosing Problems with CAD Import

A number of tools have been made available to assist in the diagnosing of problems with CAD data import. As already described and CAD data sketch can be compared with the sketch generated by the Wound Composite Modeler. Also, if an error occurs while generating the geometry of the layer, the layer gets deactivated and the plug-in continues on to the next layer. It may help to regenerate the dome geometry in question and deactivate all of the layers after the one having problems. This may help to understand what may be causing the geometry to fail such as sharp changes in the exterior of the plies.

6. Output Processing

When the plug-in generates the COPV model, it automatically generates node sets named Layer*n* along the bottom of each layer where n is the layer number. The dialog accessed from the menu item **Plug-Ins->WoundComposites->Plot** automates the process of creating path plots along the interface between layers using these node sets. The dialog is shown in the *Figure 6-1*. For each layer selected from the table, the script first details the elements only in that particular layer, then sorts the nodes in the Layer*n* node set from one end to another, and creates an x-y curve of the chosen field variable along the list of nodes. Finally, all of the x-y curves are combined into a single path plot as shown in *Figure 6 -2*.

The field output variable, step number, and frame number must be selected prior to opening up the dialog. An option to smooth out the curves is provided. The path plots can often contains significant spikes in regions where on layer overlaps another. The *Smoothing Factor* option simply smoothes out these spikes as shown in *Figure 6-3* using a smoothing factor of 10.0.

This automated path plot utility will not work as expected on models using axisymmetric shell, three-dimensional shell, and continuum shell elements.

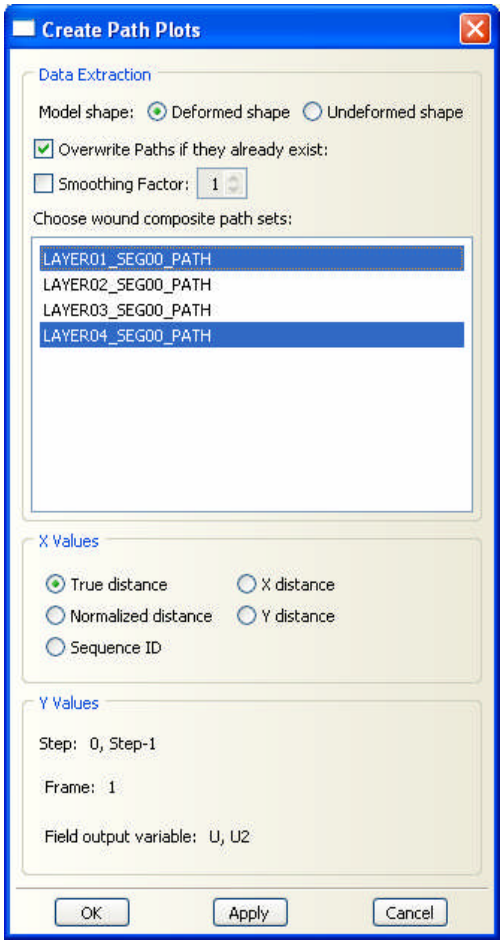


Figure 6-1: Path Plot Dialog

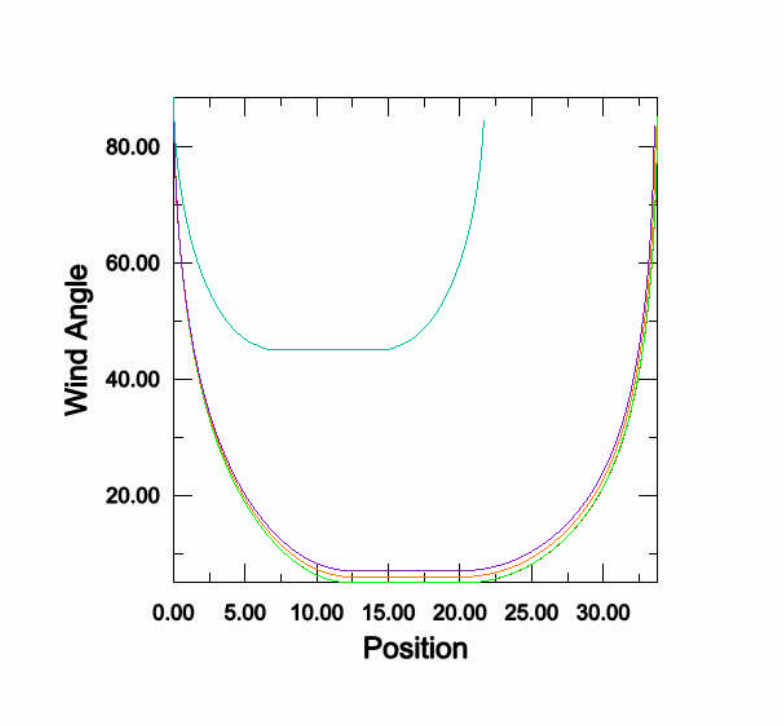


Figure 6-2: Plot of Wind Angles

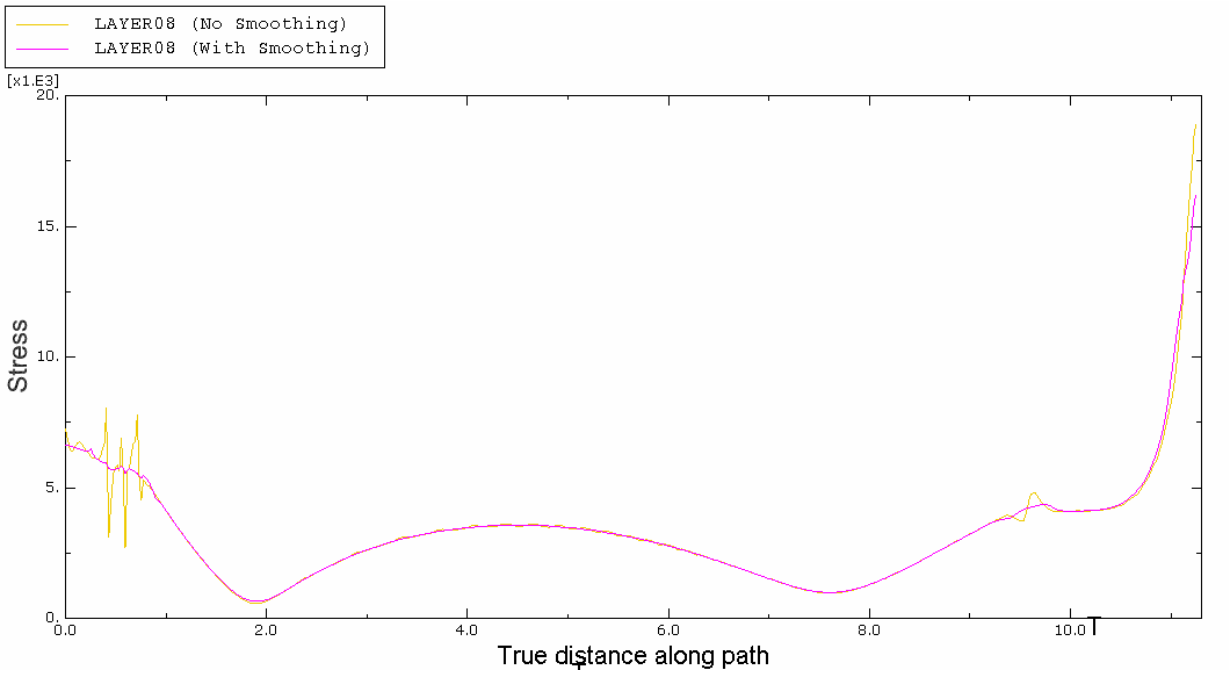


Figure 6-3: Path Plot with and without Smoothing

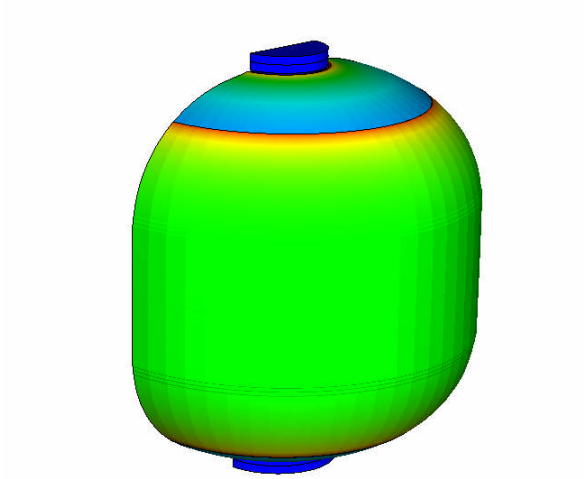


Figure 6-4: Wind Angle Contour Plot

7. Sample Test Cases

A number of QA problems have been included with the plug-in. Since these tests are fairly small but test all of the plug-in functionality, it is recommended that these be run and experimented with in order to familiarize oneself with the workings of the plug-in. Multiple tests may be selected to be run at one time. Three test modes are provided:

- Skip Material Calculations: Generates geometry and mesh only.
- Create Include Files: Generates run-ready model but does not submit the job to solver.
- Run Full Analysis: Generates and runs model. The displacements in the y-direction at the ends of tank are extracted and compared to archived results.

An Abaqus/CAE model named “QaTestModel” is created and saved of the last test chosen. Thus if a particular feature is of interest, you can scan through the test descriptions and run the test that best describes the feature of interest. The resulting model “QaTestModel” will be created and readily available for editing and testing.

An option is provided to view archived test results. This is useful when comparing baseline tests of the same model but different element types. For example, the tests in groups 4, 5, 6 and 7 are for the most part the same tests but using axisymmetric continuum geometry, 3D continuum geometry, axisymmetric shell geometry, and 3D shell geometry respectively. Selecting, for example, 4.1, 5.1, 6.1 and 7.1 then clicking the **VIEW ARCHIVED RESULTS** button will provide a comparison of results for the different element types.

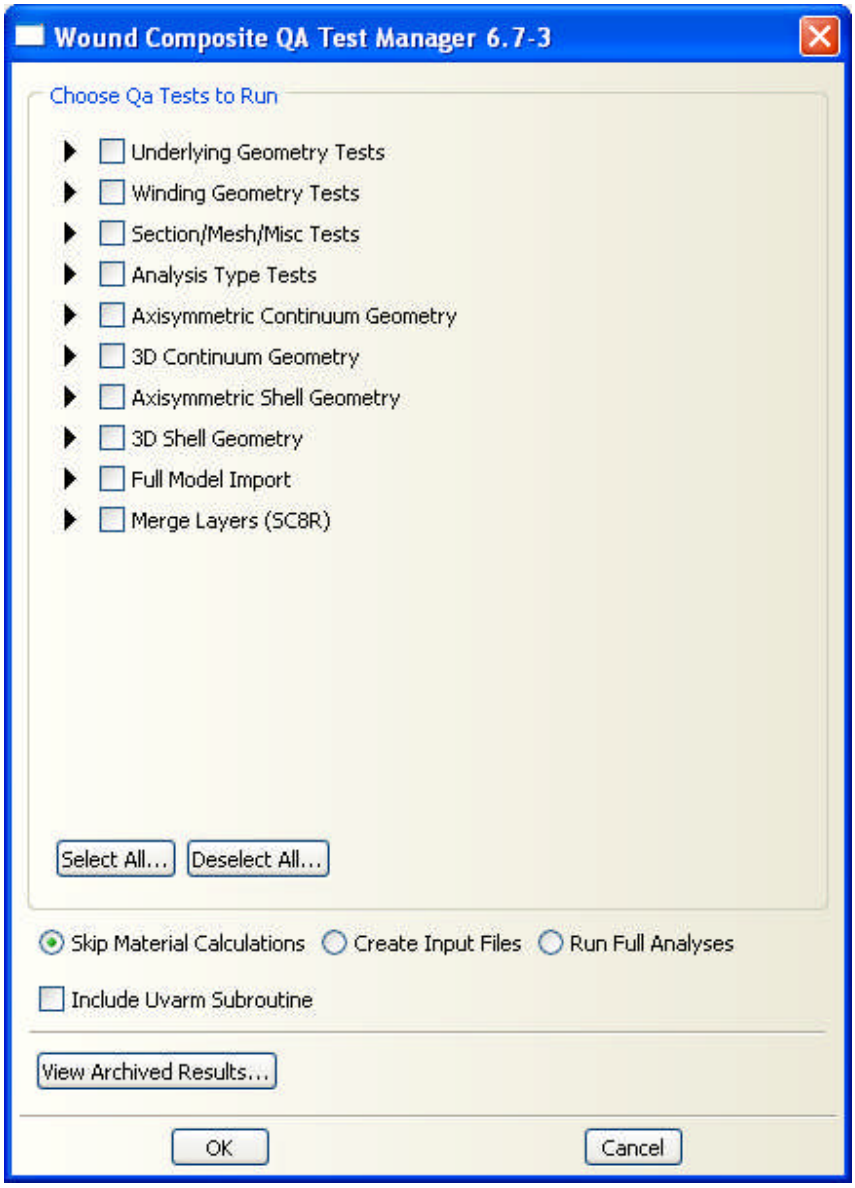


Figure 7-1 QA Test Manager Dialog

8. Installation of Plug-in

The Wound Composites Modeler for Abaqus installation disc contains a single folder named *WoundComposites* which contains all the files necessary to run the plug-in. This entire folder and all its subfolders should be installed under an *abaqus_plugins* folder as described below and should only be run in Version 6.7 of Abaqus.

When you start Abaqus/CAE, it searches for plug-in files in the following directories and all their subdirectories:

- *abaqus_dir*\cae\abaqus_plugins where *abaqus_dir* is the Abaqus parent directory.
- *home_dir*\abaqus_plugins where *home_dir* is your home directory.
- *current_dir*\abaqus_plugins where *current_dir* is the current directory.

Abaqus/CAE will import any files in these directories that match the naming convention *_plugin.py. The *abaqus_dir* directory for Windows is usually C:\Abaqus\6.7-1 for Abaqus Version 6.7-1.

Thus, if it is preferred that any user in question who has access to the Abaqus installation be allowed access to the plug-in, then the *WoundComposites* folder should be copied to the *abaqus_dir*\cae\abaqus_plugins folder. If the plug-in should be made available only on a user-by-user basis, then the *WoundComposites* folder should be copied to a folder named *home_dir*\abaqus_plugins in the user's home directory.

9. References

Papers

1 Peters, S.T., Humphrey, W.D, and Foral, R.F., Filament Winding Composite Structure Fabrication, 2nd Edition.

2 Gray, D.L., and Moser, D.J., "Finite Element Analysis of a Composite Overwrapped Pressure Vessel", American Institute of Aeronautics and Astronautics.

3 Skinner, Michael, "Trends, advances and innovations in filament winding", Reinforced Plastics, February 2006.

Abaqus References

For additional information on the Abaqus capabilities referred to in this brief please see the following Abaqus Version 6.7 documentation references:

- Analysis User's Manual
- Abaqus GUI Toolkit User's Manual
- Abaqus User's Manual